Digimat 2023.1

RP User's Guide

HEXAGON

Americas
5161 California Ave. Suite 200
University Research Park
Irvine, CA 92617
Telephone: (714) 540-8900
Email: americas.contact@hexagon.com

Europe, Middle East, Africa
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: info.europe@hexagon.com

Japan
KANDA SQUARE 16F
2-2-1 Kanda Nishikicho, Chiyoda-ku
1-Chome, Shinjuku-Ku
Tokyo 101-0054, Japan
Telephone: (81)(3) 6275 0870
Email: MSCJ.Market@hexagon.com

Asia-Pacific
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@hexagon.com

Worldwide Web
www.mscsoftware.com, www.hexagon.com

Support
https://simcompanion.hexagon.com

March 23, 2023

DIGI:V2023.1:Z:Z:Z:DC-UG-PDF

Documentation Feedback

At Hexagon, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us.

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about Hexagon documentation by taking a short 5-minute survey.

Note:	The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact Technical Support.

Contents

Digimat RP User's Guide

Preface

1 Overview

2 Prerequisites

3 Structural Model Window

4 Material Window

5 Manufacturing Data Window

6 Solution Settings Window

7 FEA Jobs Submission and Monitoring

8 Fatigue Lifetime Computation Window

9 Command Line/Batch Mode

Preface

About This Guide

This Guide is the *Digimat RP User Guide*. It contains information about the standard usage of Digimat-RP and it's interface with 3rd party software tools.

Purpose of This Guide

This guide explains the procedure for standard usage of Digimat RP. The purpose of this guide is to provide user:

- An overview of Digimat-RP capabilities for easily couple manufacturing data with a finite element analysis
- Details of the GUI of Digimat-RP and explain standard usage workflows
- Information to perform post-processing for Fatigue life calculations
- An overview for you of how to run an analysis in batch mode

The information in this manual is mostly descriptive. You will find some techniques discussed in detail.

Typographical Conventions

This section provides a brief overview of the typographical conventions used in the document to help the user better follow the Digimat documentation.

This section describes some syntax that will help you in understanding text in the various chapters and thus in facilitating your learning process. It contains stylistic conventions to denote user action, to emphasize particular aspects of Digimat to signal other differences within the text.

LM Sans 10	Body and general text
Courier New	▪ Represents command-line options of Digimat. ▪ Directory names and paths ▪ File names and Paths ▪ Linux terminal script `Example: lmreread -c <parent>/msc/MSC.Licensing/licenses/license.dat`

Bold Text	▪ Highlights ▪ Dialog box names ▪ Buttons ▪ Menus ▪ User inputs ▪ The commands/user inputs for all descriptions related to terminal commands. ▪ Default values Example: `[root@vm-tmrhel73 MSC]#` **./msc_licensing_helium_linux64.bin**
HelveticaNeue LT Pro Cn 57	▪ Hyperlinks ▪ Weblinks Example: Chapter 7: FEA Jobs Submission and Monitoring
Italic Text	Represents references to books. Example: *Digimat AM User's Guide*

Technical Support

For technical support phone numbers and contact information, please visit:
https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304

Support Center

https://simcompanion.hexagon.com

The Support Center provides technical articles, frequently asked questions, and documentation from a single location.

Accessing Digimat Documentation

This section describes how to access the Digimat documentation outside of Digimat. Digimat documentation is available through PDF files. The PDF files can be obtained from the following sources:

- Digimat documentation installer
- SimCompanion
- Combined documentation

The PDF documentation files are appropriate for viewing and printing with Adobe Acrobat Reader (version 10.1.4 or higher), which is available for most Windows and Linux systems. These files are identified by a `.pdf` suffix in their file names.

Downloading the PDF Documentation Files

You can download the PDF documentation from SimCompanion (https://simcompanion.hexagon.com).

Navigating the PDF Files

For the purpose of easier online document navigation, the PDF files contain hyperlinks in the table of contents and index. In addition, links to other guides, hyperlinks to all cross-references to chapters, sections, figures, tables, bibliography, and index entries have been applied.

To open the cross reference to other guides in a new window, you can make following changes to your Adobe Reader settings:

1. Click **Edit** -> **Preferences**.
2. Select **Documents.**
3. Uncheck **Open cross-document links in same window** option.
4. Click **OK**.

Printing the PDF Files

Adobe Acrobat PDF files are provided for printing all or part of the manuals. You can select the paper size to which you are printing in Adobe Acrobat Reader by doing the following:

1. Click **File**.
2. Select the **Print**.... option. The **Print** dialog box is displayed.
3. Select **Page Setup**....
4. Choose the required paper size in the **Page Setup** menu.

The PDF files are recommended when printing long sections since the printout will have a higher quality.

If the page is too large to fit on your paper size, you can reduce it by doing the following:

1. Select the **File** -> **Print**.
2. Under **Page Scaling,** choose the **Shrink to Printable Area** option.

Internet Resources

Hexagon (www.hexagonmi.com/mscsoftware)

Hexagon corporate site with information on the latest events, products, and services for the CAD/CAE/CAM marketplace.

Hexagon Download Center (https://mscsoftware.subscribenet.com/)

1 Overview

- Microsoft .NET prerequisites
- General description
- Digimat-RP Startup
- Main window
- Ribbon
- Settings Manager
- Color Scale Manager
- Supported Solutions Per Performance, Material and FEA Code

Microsoft .NET prerequisites

The Microsoft .NET Framework 4.8 or higher is required to use Digimat-RP. See Chapter Windows Prerequisites in the *Digimat Installation and Operations Guide* for more details.

General description

Digimat-RP is a powerful software which enables to bridge the gap between processing simulation, e.g., injection molding, and predictions carried out on the structural side, e.g., implicit & explicit structural FEA.

In short, Digimat-RP enables an integrative, accurate and efficient approach to multi-scale material and structure modeling by taking into account the process-induced material microstructure in the FEA of the final part structure.

Below is a brief description of the Digimat-RP software interfaces and its major capabilities. Please refer to the detailed sections of the documentation or contact digimat.support@hexagon.com for a demonstration or for more detailed information concerning the Digimat software suite.

Digimat-RP Startup

Digimat-RP working environment is organized around projects. When starting Digimat-RP, the user can either:

- Create a new project
- Load a recently saved project
- Load an existing project from a file

A project is defined by a name and a workflow. Once the definition of the project is done, the user can access the main window of Digimat-RP.

Several types of workflow are available:

- Generic: It has been introduced in Digimat 2022.4 to cover all standard workflows available with previous Digimat versions.
- Fatigue: It has been introduced in Digimat 2022.4 to cover all specific workflows for Fatigue.
- Launch legacy Digimat tools: It has been introduced in Digimat 2023.1 and allows to start Digimat-CAE and Digimat-MAP directly from Digimat-RP. These options do not create a project in Digimat-RP, and are equivalent to starting these tools from the Digimat platform.
- Legacy manual workflow: It has been introduced in Digimat 2023.1 to cover situations where the Digimat material coupling is done manually instead of being managed by Digimat-RP. For example, this includes cases where the model is not available or not supported by Digimat-RP. This workflow allows to only generate the interface file without input deck.

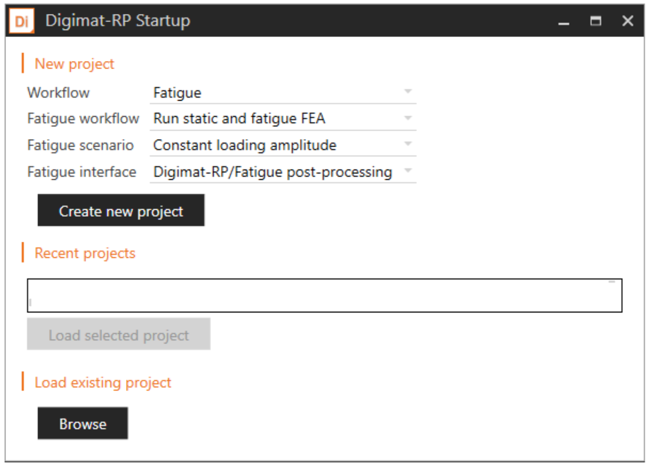

Figure 1-1 Example of the Digimat-RP startup window when selecting the workflow Fatigue.

When selecting the workflow Fatigue, following additional choices are requested to the user:

- Fatigue workflow: Workflow that the user will use to compute the lifetime.
- Fatigue scenario: Scenario for which the user wants to compute the lifetime.
- Fatigue interface: Fatigue code that the user wants to use to compute the lifetime.

Three choices are offered as Fatigue workflow:

- Run static and fatigue FEA: This means that the user wants to run the static analysis then run the Fatigue FEA computation into the same project.
- Run fatigue FEA from static FEA results: This means that the user has already run the static analysis and only wants to run the Fatigue FEA computation on the existing results file. Please be aware that some limitations in the remaining usage of RP are introduced once selecting this Fatigue workflow combined with external Fatigue interfaces. The details are explained in the chapter of each Digimat-RP window. When selecting this Fatigue workflow in combination with RP/Fatigue Post-processing, the user is directly accessing the RP/Fatigue post-processing window once the project is created.
- Run static FEA with fatigue failure criteria: This means that the user wants to run a single simulation which is static but also contains a lifetime prediction. This option will only enable the user to do Constant Loading Amplitude scenario.

Four choices are offered as Fatigue scenario:

- Constant Loading Amplitude: This means that the user wants to compute the lifetime for a simple signal for which the macroscopic load ratio is known.
- Time transient: This means that the user requires the fatigue FEA code to extract the signal from the simulation in order to compute the lifetime.
- Varying loading amplitude: This means that the user wants to compute the lifetime for a complex signal described by a file.

- Vibrational: This means that the static analysis is an NVH simulation while for all other scenarios, the static analysis is a quasi-static one.

Three Fatigue interfaces are supported:

- Digimat-RP/Fatigue Post-processing: An internal Digimat tool to compute the lifetime for Constant Loading Amplitude
- CAEfatigue: A Hexagon tool to compute the lifetime for all scenarios
- nCode: An external tool to compute the lifetime for all scenarios

Main window

The main window of Digimat-RP is divided in two parts:

- a workspace
- a ribbon bar

The workspace area can hold five different windows:

- Structural model window (which is displayed maximized at application startup)

  or

  Structural results window for the Run fatigue FEA from static FEA results workflow with CAEfatigue or nCode
- Digimat material window
- Manufacturing data window
- Solution settings window
- Fatigue post-processing window

These windows can be accessed via the relevant tab in the ribbon, or via the action buttons available in the Component workflow tree on the **Structural model** window.

The ribbon bar itself is divided into three areas, the **File group**, the **Workflow navigation** and the **Job submission controls**.

Ribbon

The ribbon enables the user to work on global aspects of the Digimat-RP project as well as to quickly navigate through all steps required to set up a coupled analysis.

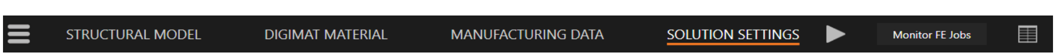

Figure 1-2 The Digimat-RP ribbon.

File Group

The **File group** contains standard functions to save and load a Digimat-RP project. It also gives access to the **Settings manager**. (see Figure 1-3).

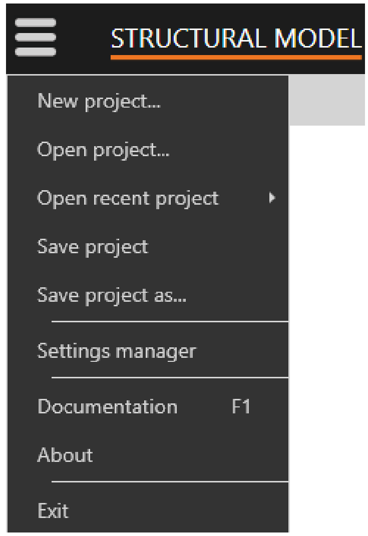

Figure 1-3 The application menu.

Digimat-RP projects can be saved in two formats:

- The Digimat complete project (`.dcp`) format, which contains a copy of the full project. Hence, the input files (input deck, manufacturing data files,...) used to create the project are not required when loading a `.dcp` file.
- The Digimat lightweight project (`.dlp`) format, which is usually much smaller than the corresponding `.dcp` file. However, in order to be able to load the project, the input files used to create the project must still be present at the same original location on the disk.

By default, both `.dcp` and `.dlp` files are associated to Digimat-RP in Windows so opening a `.dcp` or `.dlp` file will start Digimat-RP and automatically load the project. This can also be done from the command line by passing the path to a `.dcp` or `.dlp` file as first argument.

Workflow Navigation

The **Workflow navigation** contains tabs related to the four steps required in a complete workflow:

- **Structural model**: the definition of a structural model and selection of the component to assign a Digimat material and its manufacturing data

  or

  Structural results window: the definition of the results file for the Run fatigue FEA from static FEA results workflow with CAEfatigue or nCode

- **Digimat material**: the definition of a Digimat material to be applied to a component of the structural model
- **Manufacturing data**: the definition of the manufacturing data associated to the Digimat material model
- **Solution settings**: the review and adjustment of the solution settings
- **Fatigue post-processing window**: the tool to compute lifetime for Constant Loading Amplitude scenario

Each one of these steps is associated to one window in the workspace. Depending on the current step in the workflow, each tab may be available or disabled. Clicking on a tab brings the associated window to the front, in combination with the **Structural model** window which remains always present.

FE Analyses Group

When all steps of the workflow have been successfully completed for a component, the **Submit** button becomes active. Clicking on it starts the generation of all the files required to run the coupled analysis with the defined Digimat materials. After a successful generation, a table containing the description of the state variables used by Digimat during the coupled analysis can be viewed by clicking on the **State Vars** button.

Digimat-RP can also be used to launch the coupled analysis and monitor its progress. The **Monitor FE jobs** button opens the job manager which contains all coupled analyses that have been launched during this session. See section Submitting Jobs for more information about jobs.

Depending on the selected workflow, several (or all) of these buttons may be unavailable. For example, in workflows where there is no job launched and managed by Digimat-RP, the **Monitor FE jobs** button is absent.

Settings Manager

Many different settings of Digimat-RP can be customized using the **Settings manager**. It can be accessed through the File menu.

All these settings are permanently saved when clicking on **Apply**, or by closing the manager with **OK**. Clicking on **Cancel** closes the manager without saving any modification. It is also possible to import and export GUI settings.

Note that the settings contained in the **Settings manager** are kept even after uninstalling and reinstalling Digimat-RP. After a reinstallation, clicking on **Restore all defaults** allows to reset all settings to their default values if the previous ones should be discarded.

The **Settings manager** window is divided into five different tabs detailed hereafter.

Executable File Paths

The first tab contains all settings related to executables required to locally run a job. If a setting is missing, then it will be impossible to run related analyses and/or to open the results files after completion of the job. Of course, it will still be possible to generate the coupled analysis files and manually launch the job, either locally or on another computer.

There are three ways to modify these settings:

- by entering the full path to the executable file in the text box
- by browsing the file on the disk
- by using an auto-detection function

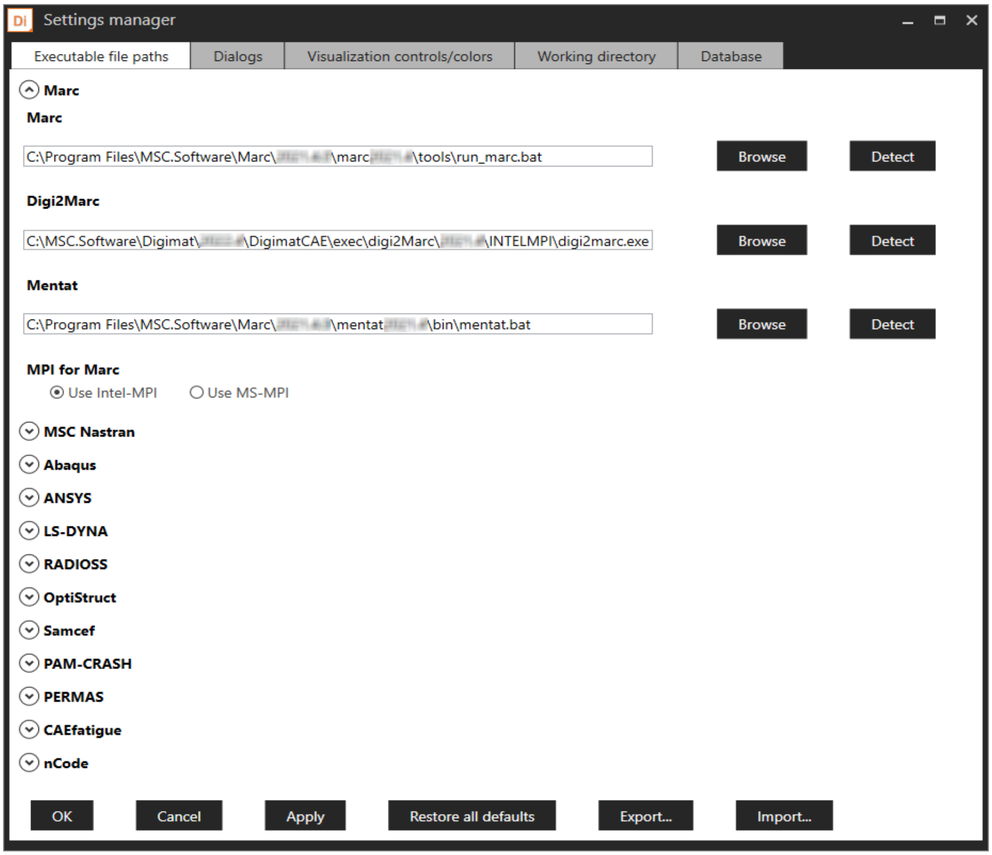

Figure 1-4 The Executable file paths tab.

The auto-detection only looks into the standard installation paths (e.g., `C:\Program Files\MSC.Software\MSC_Nastran` for MSC Nastran), standard environment variables and the paths given during the installation of Digimat or defined in the DIGIMAT_Settings.ini File from *Digimat Installation and Operations Guide*. Thus, it may not detect all executables. If several files are detected, a window showing all possibilities will be displayed so that the correct version can be chosen, as shown in Figure 1-5.

Note that if one or several executable file paths are missing when starting Digimat-RP, it will automatically try to detect them during the startup.

Table 1-1 shows the executable files required to locally run coupled analyses and Table 1-2 shows usual locations of the files.

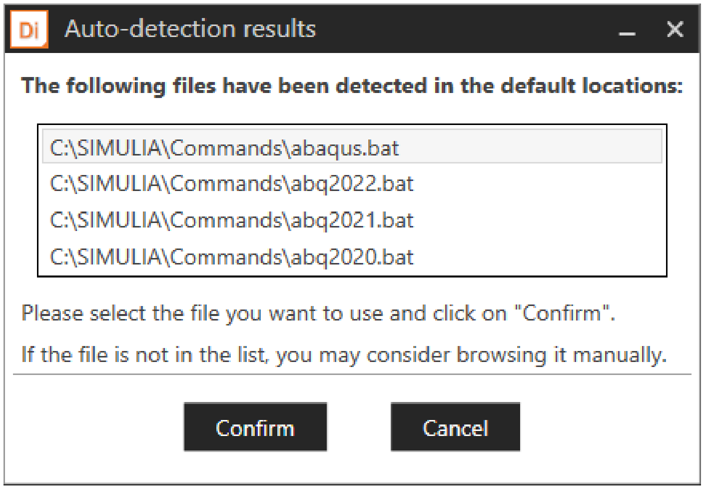

Figure 1-5 Auto-detection of the path to Abaqus.

Table 1-1 Executable file paths settings: required executable files.

Setting	Required For	Accepted Executable File Name
Marc	Marc (run)	`run_marc.bat`
Digi2Marc	Marc (run)	`digi2marc*.exe, marc*.exe`
Mentat	Marc (results)	`mentat.bat`
MSC Nastran	MSC Nastran (run)	`nastran.exe, nast*.exe`
Patran	MSC Nastran (results)	`patran.exe`
Abaqus	Abaqus (run, results)	`abaqus.bat, abq*.bat, abq*.exe`
ANSYS	ANSYS (run, results)	`ansys*.exe`
LS-DYNA	LS-DYNA (run)	`ls*.exe, mpp*.exe`
LS-PrePost	LS-DYNA (results)	`lsprepost.exe`
OptiStruct	OptiStruct (run)	`optistruct.bat`
HyperWorks	OptiStruct (results)	`hw.exe`
Samcef	Samcef (run, results)	`samcef.cmd`
Digi2Samcef	Samcef (run)	`my_Mecano.exe, my_Dynam.exe`
PAM-CRASH	PAM-CRASH (run)	`pamcrash.bat`
Visual-Viewer	PAM-CRASH (results)	`VisualEnvironment.bat`

Table 1-1 Executable file paths settings: required executable files.

Setting	Required For	Accepted Executable File Name
PERMAS	PERMAS (run)	permas.bat
MPI	LS-DYNA (run)	mpirun.exe, mpiexec.exe, impiexec.exe
CAEfatigue	CAEfatigue	ProcessApp.exe
nCode	nCode	ncodedesktop.exe

Table 1-2 Executable file paths settings: standard location of the executable files. The series of X's denote the version numbers. Several variants not mentioned here are also taken into account.

Setting	Standard Location
Marc	C:\MSC.Software\Marc\XXXX\marcXXXX\tools\run_marc.bat C:\Program Files\MSC.Software\Marc\XXXX\marcXXXX\tools\run_marc.bat
Digi2Marc	No default location, see section Digimat-CAE/Marc
Mentat	C:\MSC.Software\Marc\XXXX\mentatXXXX\bin\mentat.bat C:\Program Files\MSC.Software\Marc\XXXX\mentatXXXX\bin\mentat.bat
MSC Nastran	C:\MSC.Software\MSC_Nastran\XXXXX\bin\nastran.exe C:\Program Files\MSC.Software\MSC_Nastran\XXXXX\bin\nastran.exe
Patran	C:\MSC.Software\Patran_x64\XXXXX\bin\patran.exe C:\Program Files\MSC.Software\Patran_x64\XXXXX\bin\patran.exe
Abaqus	C:\SIMULIA\Commands\abaqus.bat
ANSYS	C:\Program Files\ANSYS Inc\vXXX\ansys\bin\winx64\ANSYSXXX.exe
LS-DYNA	No default location, see section Digimat-CAE/LS-DYNA
LS-PrePost	C:\Program Files\LSTC\LS-PrePost\XXX-X64\lsprepost.exe
OptiStruct	C:\Program Files\Altair\XXXX\hwsolvers\scripts\optistruct.bat
HyperWorks	C:\Program Files\Altair\XXXX\hw\bin\win64\hw.exe
Samcef	C:\Program Files\Samtech\Samcef\VXXXXXXX_x64\exec\samcef.cmd
Digi2Samcef	No default location, see section Digimat-CAE/Samcef
PAM-CRASH	C:\Program Files\ESI Group\Virtual-Performance\XXXXXX\Solver\bin\pamcrash.bat
Visual-Viewer	C:\Program Files\ESI Group\Visual-Environment\XXX\Windows-x64\VisualEnvironment.bat

Table 1-2 Executable file paths settings: standard location of the executable files. The series of X's denote the version numbers. Several variants not mentioned here are also taken into account.

Setting	Standard Location
PERMAS	`C:\Program Files\INTES\VXX\bin\permas.bat`

MPI	`C:\Program Files (x86)\Platform Computing\Platform-MPI\bin\mpirun.exe`
	 `C:\Program Files\Microsoft HPC Pack XXXX\Bin\mpiexec.exe`
	 `C:\Program Files\Intel MPI XXXX\x64\impiexec.exe`

CAEfatigue	`C:\Program Files\MSC.Software\CAEfatigue\XXXX\Windows\ProcessApp.exe`
nCode	`C:\Program Files\nCodeXXXX\bin\ncodedesktop.exe`

Dialogs

The **Dialogs** tab contains GUI preferences settings related to dialogs and messages that may appear in Digimat-RP, and whether automatic actions should be taken.

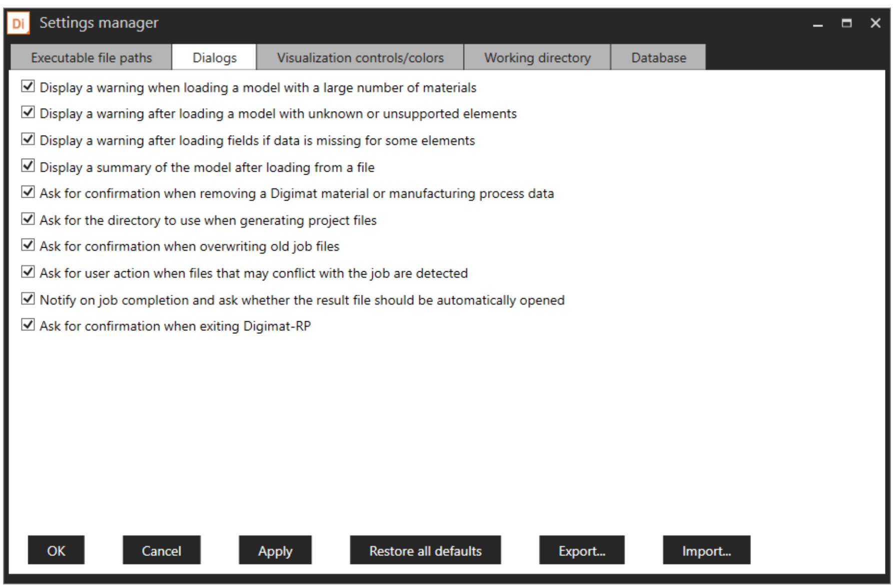

Figure 1-6 The Dialogs tab.

Visualization Controls/Colors

The **Visualization controls/colors** tab contains GUI settings related to the model visualizer used to show the structural model and the manufacturing process model (if there is one). The colorbar palette and the behavior of mouse buttons (select, rotate, pan or zoom) can also be changed here.

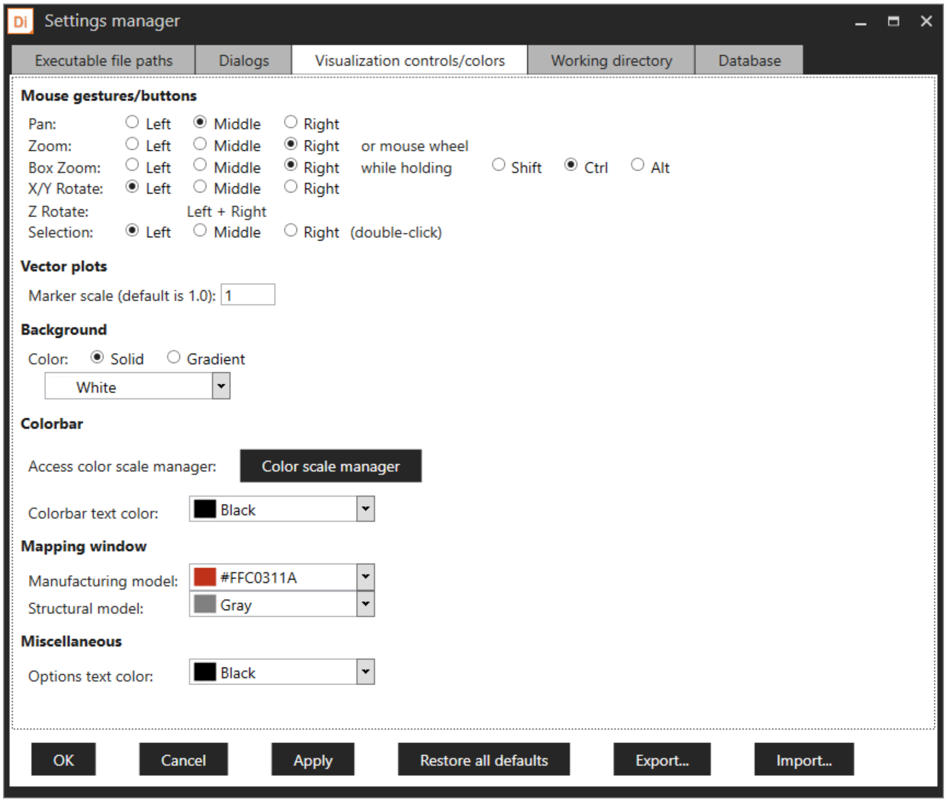

Figure 1-7 The Visualization controls/colors tab.

Working Directory

This tab allows you to change the working directory of Digimat-RP. This setting influences the location of the temporary files produced by Digimat-RP (which are deleted when exiting the application) and the default directories used to produce the project files. The actual directory used by each project can still be customized when submitting the analysis.

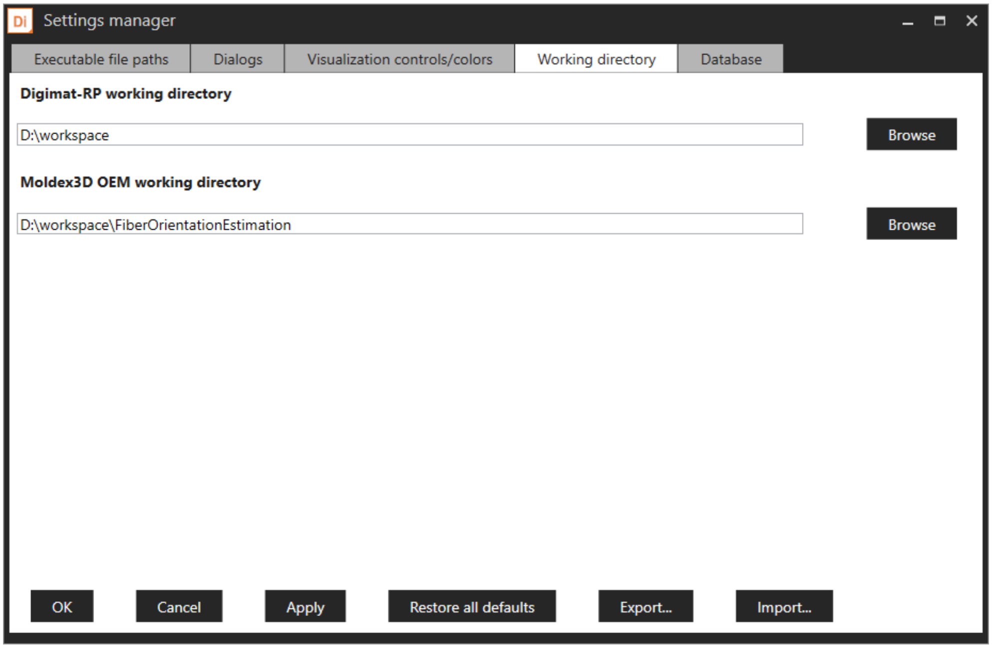

Figure 1-8 The Working directory tab.

Database

Digimat-RP supports access to both local and remote databases. The database connection settings can be set in this tab as shown in Figure 1-9 and it is possible to switch between a local and a remote database at any time. When using a remote database, all private materials created and shared by other users will also be accessible in Digimat-RP.

When opening Digimat-MX from Digimat-RP, these database connection settings will be automatically used to establish the connection to the database.

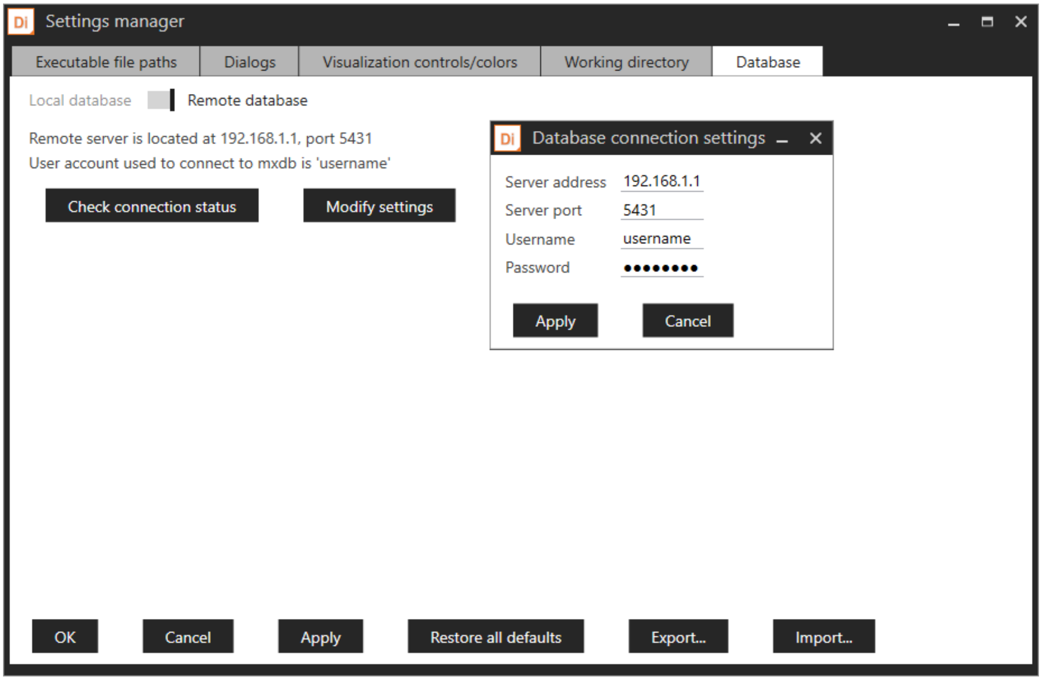

Figure 1-9 Database connection settings for a remote database.

Color Scale Manager

The color scale used during post-processing can be fully customized by using the **Color scale manager**. The latter is accessible by clicking on the color scale on the right-hand side of the screen, or through the **Settings manager**, in the **Visualization** tab.

The **Color scale manager** (see Figure 1-10) offers the possibility to:

- Change the colors used by the color scale (color type and number of colors), including for out-of-range values;
- Use a continuous (Figure 1-11) or discrete (Figure 1-12) color scale;
- Define the minimum and maximum values of the color scale;
- Save a color scale in the manager for later use. The color scale becomes accessible from the drop-down list located on the right;
- Export a color scale to a file (`.dcs` file). Importing `.dcs` files is also possible;
- Save a color scale as default. This scale will be used by default in future sessions of Digimat-RP;
- Change the font size of the color bar. This is useful to make the text more visible for snapshots.

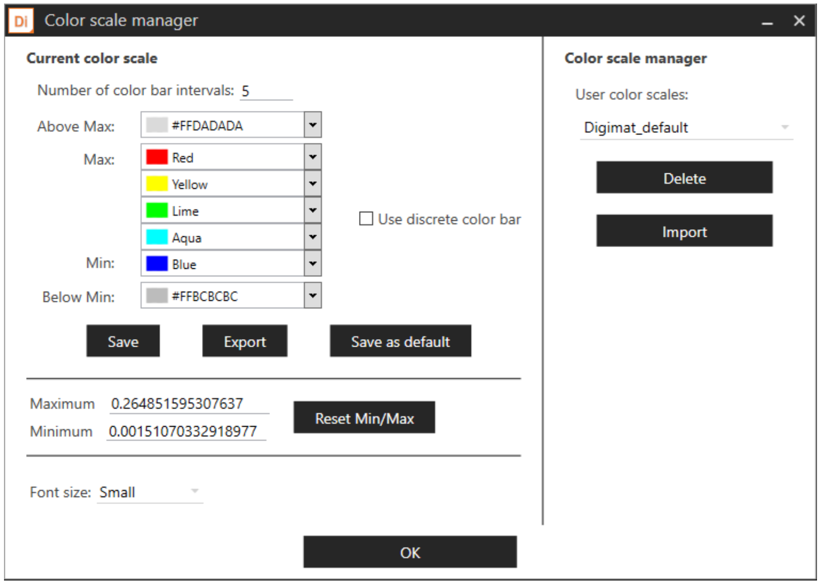

Figure 1-10 Color scale manager.

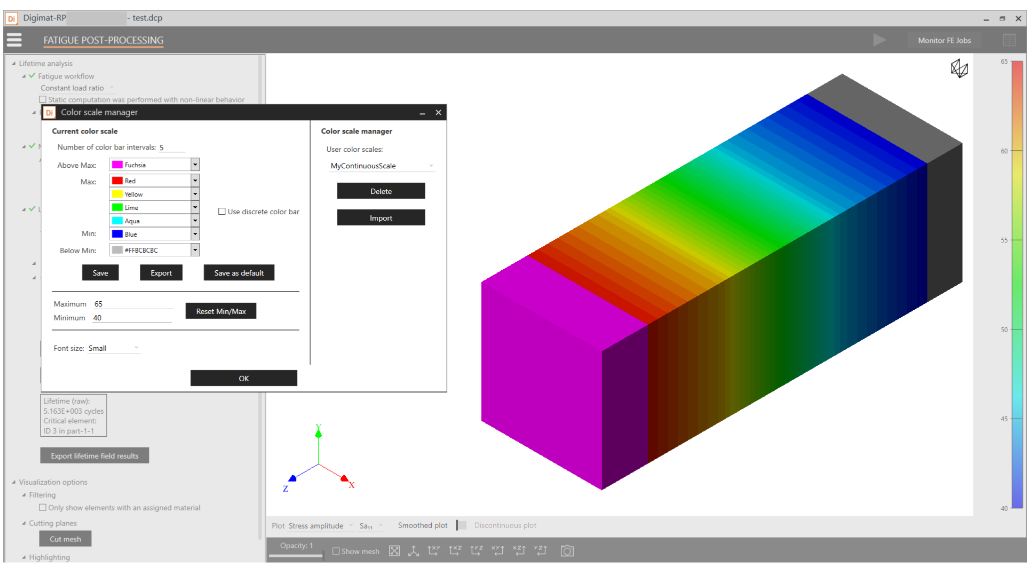

Figure 1-11 Example of a continuous custom color scale.

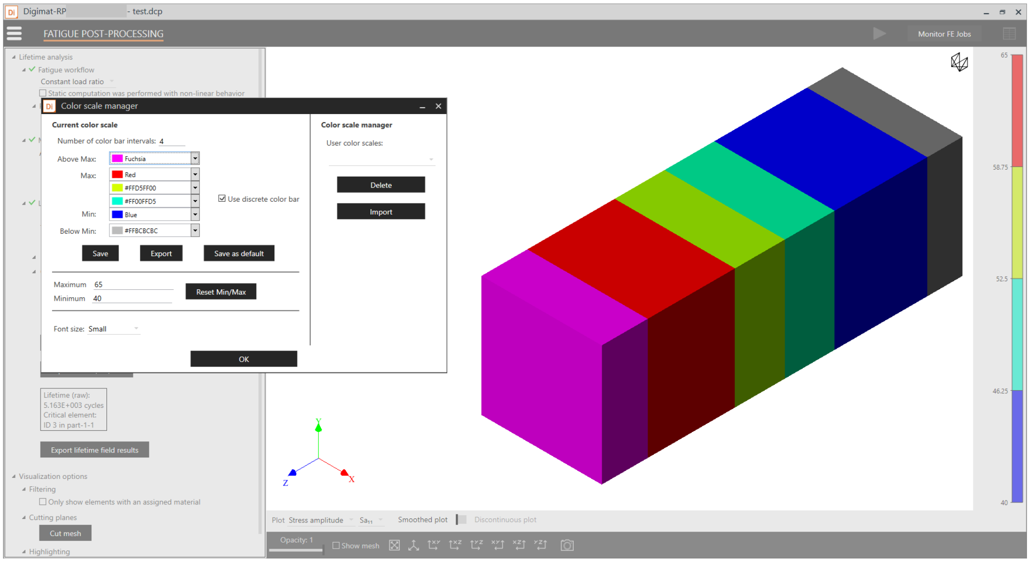

Figure 1-12 Example of a discrete custom color scale.

Supported Solutions Per Performance, Material and FEA Code

The following table shows the supported solutions depending on the material type, the process, the FEA code, the performance and the matrix material model. The table is restricted to 2-phases composites with the exception of mucell. The default solution is also displayed.

Table 1-3 Supported solutions per performance, material and FEA code

Material Type	Process	FEA Code	Performance	Material Model	Available Solutions	Default Solution
SFRP, LFRP	IM	Abaqus/Exp, Abaqus/Std, ANSYS, LS-DYNA/Imp, Marc, Nastran/SOL400, PAM-CRASH, Samcef	Crash	(V)EVP / SREP + FPGF	Hybrid	Hybrid
SFRP, LFRP	IM	LS-DYNA/Exp	Crash	EVP / SREP + FPGF	Hybrid, Macro (Nonlinear)	Hybrid
SFRP, LFRP	IM	LS-DYNA/Exp	Crash	VEVP + FPGF	Hybrid	Hybrid
SFRP, LFRP	IM	Abaqus/Std, ANSYS, LS-DYNA/Imp, Marc, Samcef	Creep	(T)VE	Micro, Hybrid	Hybrid
SFRP, LFRP	IM	Abaqus/Std, ANSYS, LS-DYNA/Imp, Marc, Samcef	Creep	(T)EVP, (T)SREP	Hybrid	Hybrid
SFRP, LFRP	IM	Nastran/SOL400	Creep	VE	Micro, Hybrid	Hybrid
SFRP, LFRP	IM	Nastran/SOL400	Creep	EVP, SREP	Hybrid	Hybrid

Material Type	Process	FEA Code	Performance	Material Model	Available Solutions	Default Solution
SFRP, LFRP[1]	IM	Abaqus/Std, ANSYS, LS-DYNA/Imp, Marc, Nastran/SOL400, Samcef	Fatigue	(V)E + PG	Micro, Hybrid	Hybrid
SFRP, LFRP[2]	IM	Abaqus, ANSYS[4], LS-DYNA/Exp	Linear stiffness	Elasticity	Micro, Hybrid, Macro (Linear)	Hybrid
SFRP, LFRP[2]	IM	ANSYS[5], LS-DYNA/Imp, Marc, Nastran/ Sol400, PAM-CRASH, Samcef	Linear stiffness	Elasticity	Micro, Hybrid, Macro (stf-dsf)	Macro (Linear)
SFRP, LFRP[2]	IM	Nastran/SOL1XX, OptiStruct, PERMAS	Linear stiffness	Elasticity	Macro (Linear)	Hybrid
Unfilled, fiber reinforced	FFF/FDM	Abaqus/Std, Abaqus/Exp, ANSYS, LS-DYNA/Exp, LS-DYNA/Imp, Marc, Nastran/SOL400, PAM-CRASH, Samcef	Linear stiffness, non-linear stiffness (+failure)	Elasticity, J_2-plasticity, Drucker-Prager (+failure)	Hybrid	Hybrid
Unfilled, fiber reinforced	SLS	Abaqus/Std, Abaqus/Exp, ANSYS, LS-DYNA/Exp, LS-DYNA/Imp, Marc, Nastran/SOL400, PAM-CRASH, Samcef	Linear stiffness, non-linear stiffness (+failure)	Elasticity, J_2-plasticity, Drucker-Prager (+failure)	Micro, Hybrid	Hybrid
SFRP, LFRP[2]	IM	Abaqus/Std, ANSYS[4]	Linear thermomechanical	TE	Micro, Hybrid, Macro (Linear)	Hybrid
SFRP, LFRP[2]	IM	ANSYS [5], LS-DYNA/Imp, Marc, Samcef	Linear thermomechanical	TE	Micro, Hybrid, Macro (stf-dsf)	Hybrid
SFRP, LFRP[2]	IM	Nastran/SOL1XX, OptiStruct, PERMAS	Linear thermomechanical	TE	Macro (Linear)	Macro (Linear)
SFRP, LFRP	IM	Abaqus/Exp, Abaqus/Std, ANSYS, LSDYNA/ Imp, Marc, Nastran/SOL400, PAM-CRASH	Non-linear stiffness	J_2-plasticity, Drucker-Prager	Micro, Hybrid	Hybrid
SFRP, LFRP[2]	IM	ANSYS [4], LS-DYNA/Exp	Non-linear stiffness	J_2-plasticity	Micro, Hybrid, Macro (Non-linear)	Hybrid
SFRP, LFRP	IM	ANSYS, LS-DYNA/Imp, LSDYNA/ Exp	Non-linear stiffness	Drucker-Prager	Micro, Hybrid	Hybrid
SFRP, LFRP	IM	Abaqus/Exp, Abaqus/Std, ANSYS, LSDYNA/ Imp, Marc, Nastran/SOL400, PAM-CRASH	Non-linear stiffness + failure	J_2-plasticity / Drucker-Prager + FPGF	Hybrid	Hybrid
SFRP, LFRP [2]	IM	LS-DYNA/Exp	Non-linear stiffness + failure	J_2-plasticity + FPGF	Hybrid, Macro (Nonlinear)	Hybrid
SFRP, LFRP	IM	ANSYS[4]	Non-linear thermomechanical	TEP	Hybrid, Macro (Nonlinear)	Hybrid

Material Type	Process	FEA Code	Performance	Material Model	Available Solutions	Default Solution
SFRP, LFRP	IM	Abaqus/Std, ANSYS, LS-DYNA/Imp, Marc, Samcef	Non-linear thermomechanical (+failure)	TEP (+FPGF)	Hybrid	Hybrid
SFRP, LFRP	IM	Abaqus/Std, Marc, Nastran/SOL400, Samcef	NVH	VE	Micro, Hybrid	Hybrid
SFRP, LFRP	IM	ANSYS(5)	NVH	VE	Micro, Hybrid	Hybrid
SFRP, LFRP(2)	IM	Nastran/SOL1XX	NVH	VE	Macro (Linear)	Macro (Linear)
UD, woven(6)(7)	CFRP(8)	Abaqus/Std, Marc	Thermomechanical with curing	TVE	Hybrid, Micro	Hybrid
UD, woven(6)(7)	CFRP(8)	ANSYS	Thermomechanical with curing	TVE	Hybrid	Hybrid
UD, woven(6)(7)	CFRP(8)	Abaqus/Std, Marc	Thermomechanical with curing	TE	Micro	Micro
SFRP	IM	Abaqus/Std, Marc	Thermomechanical with curing	TVE, TE	Micro	Micro
UD	CFRP(8)	Abaqus/Exp, Marc	Linear stiffness, non-linear stiffness (+ Camanho failure)	Elasticity, J_2-plasticity (+failure)	Micro	Micro
SFRP	IM	Abaqus/Std, Marc	Thermal	Fourier	Micro	Micro
UD, woven(6)	CFRP(8)	Abaqus/Std, Marc	Thermal	Fourier	Micro	Micro
UD(9)	CFRP(8)	Marc	Coupled thermal-mechanical	Fourier TE, Fourier TVE	Micro	Micro
UD(9)	CFRP(8)	Marc	Coupled thermal-mechanical with crystallinity	Fourier TE, Fourier TVE	Micro	Micro
UD, woven(6)(7)	CFRP(8)	Abaqus/Std, Marc	Coupled thermomechanical with curing	Fourier TVE, Fourier TE	Micro	Micro
SFRP	IM	Abaqus/Std, Marc	Coupled thermomechanical with curing	Fourier TVE, Fourier TE	Micro	Micro
SMC	CM	ANSYS, Abaqus/Std, Marc, Abaqus Exp, LSDYNA Imp, LSDYNA Exp, Nastran SOL400, Pamcrash, Permas, Samcef	Linear stiffness + Failure	E, VE	Hybrid	Hybrid
SMC(10)	CM	Abaqus/Std, Marc	Thermomechanical with curing/ crystallinity	TE, TVE	Micro	Micro
SMC(10)	CM	Abaqus/Std, Marc	Coupled thermal-mechanical with curing/crystalinity	Fourier TE, Fourier TVE	Micro	Micro

(1) Available only when fiber OT is selected as manufacturing data
(2) Macro available only when fiber OT is selected as manufacturing data
(3) No damping
(4) Macro available for solid elements only
(5) Macro available for shell elements only
(6) Balanced and unbalanced basic woven only, advanced woven are not supported
(7) Curing available for solid elements only
(8) Available only with pre-defined fixed orientations
(9) Coupled thermal-mechanical analyses available for solid elements only
(10) Thermomechanical and coupled thermal-mechanical analyses available for solid elements only

In the legacy manual workflow for generation of interface file, in some combinations, additional solution types may be available. The Macro solution is not available in that workflow.

2 Prerequisites

- Prerequisites

Prerequisites

In order to take full advantage of Digimat-RP, it is strongly recommended to ensure that the structural FEA model and the Digimat material satisfy the following prerequisites:

- The structural FEA model must be complete and ready to run with the targeted FEA code before going to Digimat-RP.
- The regions of the structural FEA model where a Digimat material is to be used should have a specific material assigned to them in the original structural FEA model. If several different Digimat materials are to be used, then each associated region should have its own unique material.

  For models using CFF (Continuous Fiber Fabrication), only one single region needs to be created for both the short fiber and the continuous fiber-reinforced Digimat materials. The classification between both materials will be done automatically during the preparation of the coupled model.
- When using CFRP process, the regions of the structural FEA model where a unidirectional or woven-reinforced Digimat material is to be used should have an adequate coordinate system and/or orientation pre-defined in the original structural FEA model. Digimat-RP will not modify these orientations when creating the Digimat-coupled model. Information about how to set up these orientations can be found at Chapter 3: Coupling to Finite Element Software in *Digimat-CAE User's Guide*.
- If it is intended to perform the coupled simulation on the local machine (i.e., the one where Digimat-RP is installed), the targeted FEA code should be properly installed and configured in order to run coupled Digimat analyses (e.g., environment variables for licensing or parallel computation configuration should be set).

3 Structural Model Window

- Structural Model
- Component Selection
- Manual Workflow
- Structural Results

Structural Model

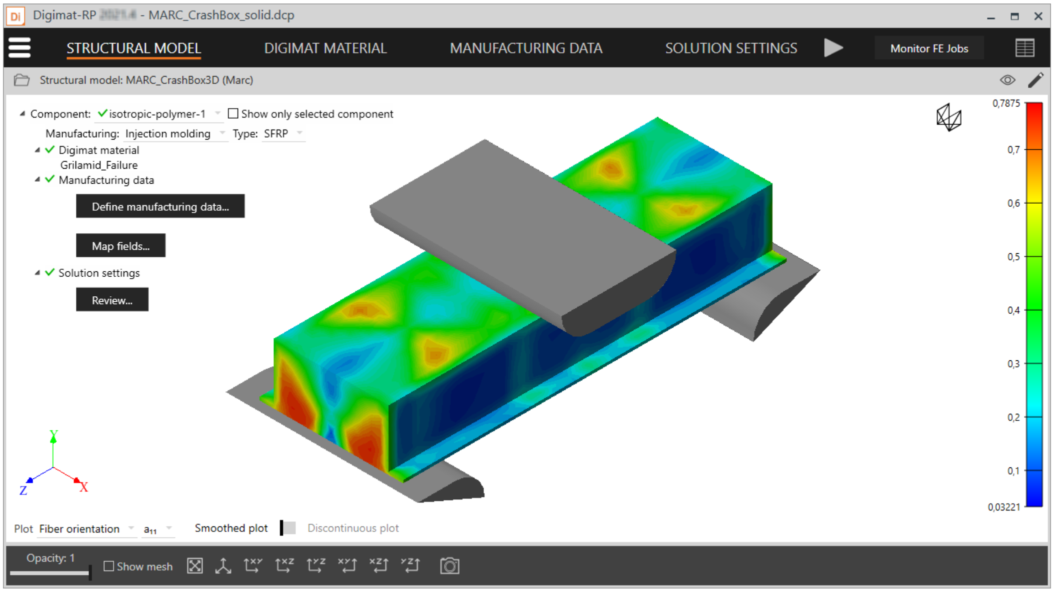

Figure 3-1 A Structural model window at the end of the workflow.

The Structural model window is the main window of the workspace, it contains a 3D visualization of the model (with available manufacturing data), and the list of components available in the structural model. A component in Digimat-RP is defined as a region of the FEA model which is assigned to a given material model. The component selection is the starting point of the Digimat-RP workflow. Upon selection of the desired component, an indication of its manufacturing type (e.g., injection molding, selective laser sintering...) and material type (e.g., unfilled polymer, short fiber reinforced polymer...) is needed. This information allows the rest of the Digimat-RP workflow to be tailored to the application and therefore to offer the best user experience.

Supported configurations in Digimat-RP 2023.1 are:

- injection molding: short fiber (SFRP) and long fiber (LFRP) reinforced polymers
- compression molding: short fiber (SFRP) and long fiber (LFRP) reinforced polymers and sheet molding compound (SMC)
- fused filament fabrication: unfilled and fiber reinforced polymers with or without lattice
- fused deposition modeling: unfilled and fiber reinforced polymers with or without lattice (Stratasys materials only)
- selective laser sintering: unfilled, bead and fiber reinforced polymers
- continuous fiber fabrication: fiber reinforced polymers and continuous fiber-reinforced polymer.

  There is no selection of the material type in this case since both polymers are required.
- CFRP process: unidirectional and woven-reinforced materials.

For each component, the workflow to set up the coupled analysis is ensured via the dedicated action buttons.

Each action button brings the relevant window in parallel to the Structural model window. When a step of the workflow is completed, it is flagged with a green check mark. When the workflow is completed for a full component, then the component name in the component list is also indicated as validated.

Supported CAE Codes and Input File Formats

Solid and shell elements are supported for the structural model. It is however possible to load CAE input files containing unsupported elements. In that case, all supported elements will be loaded and displayed and all unsupported elements will be skipped. It will then be possible to assign Digimat material to any supported element. When generating the coupled analysis files at the end of the workflow, unsupported elements will be copied without modification.

For each supported CAE code, the detailed list of supported elements is the following.

- Marc (`*.dat`)
    - Tetra 4: element type 134, 135, 157
    - Tetra 10: element type 127, 130, 133, 184
    - Pyra 5: element type 216, 217
    - Pyra 13: element type 218, 219
    - Wedge 6: element type 136, 137, 192
    - Wedge 15: element type 202, 203, 205
    - Hexa 8: element type 7, 43, 84, 113, 117, 120, 123, 188
    - Hexa 20: element type 21, 35, 44, 57, 71
    - Tri 3: element type 6, 37, 50, 138, 155, 158, 196
    - Tri 6: element type 49, 197, 200
    - Quad 4: element type 11, 18, 39, 75, 80, 85, 139, 140, 198
    - Quad 8: element type 22, 30, 72, 86, 199
- MSC Nastran (`*.bdf, *.dat, *.nas`)
    - Tetra 4 and 10: CTETRA elements
    - Pyra 5 and 13: CPYRAM elements
    - Wedge 6 and 15: CPENTA elements
    - Hexa 8 and 20: CHEXA elements
    - Tri 3: CTRIA3 and CTRIAR elements
    - Tri 6: CTRIA6 elements
    - Quad 4: CQUAD4 and CQUADR elements
    - Quad 8: CQUAD8 elements
- Abaqus (`*.inp`)

  - Tetra 4: *C3D4* elements
  - Tetra 10: *C3D10* elements
  - Pyra 5: *C3D5* elements
  - Wedge 6: *C3D6* elements
  - Wedge 15: *C3D15* elements
  - Hexa 8: *C3D8* elements
  - Hexa 20: *C3D20* elements
  - Tri 3: *S3* and *STRI3* elements
  - Tri 6: *STRI6* elements
  - Quad 4: *S4* elements
  - Quad 8: *S8* elements
- Ansys (`*.cdb, *.inp, *.dat`)
  - Hexa 8 (possibly degenerated to Wedge 6, Pyra 5 or Tetra 4): element type 5, 45, 62, 64, 65, 70, 96, 97, 164, 185, 215, 278
  - Hexa 20 (possibly degenerated to Wedge 15, Pyra 13 or Tetra 10): element type 90, 122, 186, 216, 217, 226, 231, 236, 239, 279
  - Tetra 4: element type 285
  - Tetra 10: element type 87, 92, 98, 123, 168, 187, 227, 232, 237, 240
  - Quad 4 (possibly degenerated to Tri 3): element type 41, 43, 63, 143, 157, 163, 181
  - Quad 6 (possibly degenerated to Tri 6): element type 281

  Remark: Other degenerations of Hexa elements are not supported in Digimat-RP.
- LS-DYNA (`*.k, *.key, *.dyn`)
  - Tetra 4: all
  - Tetra 10: all
  - Wedge 6: all
  - Hexa 8: all
  - Tri 3: all
  - Tri 6: all
  - Quad 4: all
  - Quad 8: all
- OptiStruct (`*.fem`)
  - Tetra 4 and 10: CTETRA elements
  - Pyra 5 and 13: CPYRAM elements
  - Wedge 6 and 15: CPENTA elements
  - Hexa 8 and 20: CHEXA elements

- Tri 3: CTRIA3 and CTRIAR elements
- Tri 6: CTRIA6 elements
- Quad 4: CQUAD4 and CQUADR elements
- Quad 8: CQUAD8 elements

■ Samcef (`*.dat`)

- Tetra 4: all
- Tetra 10: all
- Wedge 6: all
- Wedge 15: all
- Hexa 8: all
- Hexa 20: all
- Tri 3: all
- Tri 6: all
- Quad 4: all
- Quad 8: all

■ PAM-CRASH (`*.pc`)

- Tetra 4: TETR4 and SOLID elements
- Tetra 10: TETRA and TETR10 elements
- Wedge 6: PENTA6 and SOLID elements
- Wedge 15: PENT15 elements
- Hexa 8: SOLID elements
- Hexa 20: HEXA20 elements
- Tri 3: SHELL elements
- Quad 4: SHELL elements

■ PERMAS (`*.uci`)

- Tetra 4: TET4 elements
- Tetra 10: TET10 elements
- Pyra 5: PYRA5 elements
- Wedge 6: PENTA6 elements
- Wedge 15: PENTA15 and PENTA18 elements
- Hexa 8: HEXE8 and HEXFO8 elements
- Hexa 20: HEXE20 and HEXE27 elements
- Tri 3: SHELL3, TRIA3, TRIA3K, TRIA3S, TRIM3 and TRIMS3 elements
- Tri 6: TRIM6 and TRIMS6 elements
- Quad 4: SHELL4, QUAD4, QUAD4S, QUAM4, QUAMS4 and SHEAR4 elements

- Quad 8: QUAM8, QUAMS8, QUAM9 and QUAMS9 elements

Membrane elements added on the surface of solid meshes in order to obtain surface stresses are currently only supported with MSC Nastran/SOL1XX and OptiStruct. For these cases, the orientation tensor of the parent solid element is properly projected onto each membrane element.

Unit System

When a new structural model is loaded, a pop-up window is shown so that the unit system used for this model can be defined. This unit system will be used to ensure consistency between the structural model and the material models. These material models will be converted if necessary. The unit system can also be changed after loading the structural model. In this case, all loaded material models will be automatically converted to the new unit system. However, this unit system modification is only related to Digimat-RP, and will not modify the associated input deck.

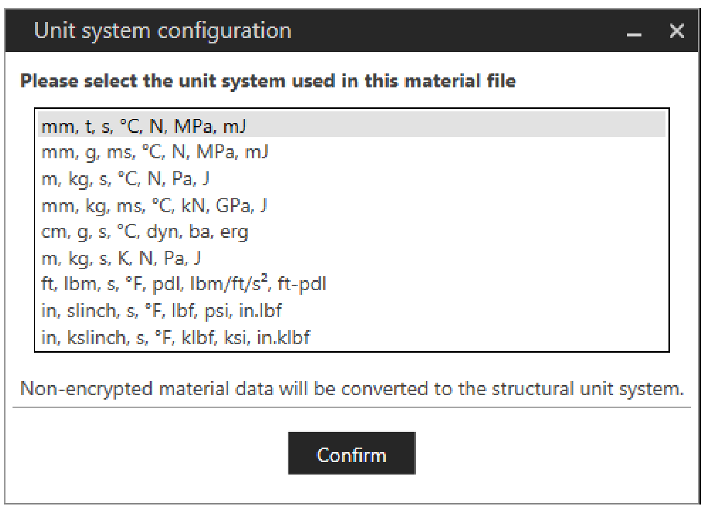

Figure 3-2 Selecting the unit system to use in Digimat-RP.

If the input deck contains a definition of unit system that is supported (see Figure 3-2), the corresponding unit system will be automatically selected. It is still possible to manually modify the unit system afterwards via the top right icon on the Structural window.

Included Files

If the input deck includes other files, these files will also be processed Digimat-RP: the cards to be modified by Digimat-RP will be modified in-place, and the cards to be added by Digimat-RP will be added next to relevant cards, in the corresponding file. The included files will be outputted at the corresponding relative location w.r.t. the project directory selected in Digimat-RP, except for files that were included using an absolute path, or a ".." token, which will be recreated at the project directory level. Table 3-1 shows the correspondence between files read and written by Digimat-RP.

For a list of limitations related to included files, refer to section Included Files.

Table 3-1 Correspondence between input files read by Digimat-RP and the coupled input deck written by Digimat-RP.

Original File Location	Output File Location
Main input deck: <input_dir>\main.dat	<project_dir>\main_DigimatCoupled.dat
File included using a filename: <input_dir>\included.dat	<project_dir>\included.dat
File included using an absolute path to a file: C:\include_dir\included.dat	<project_dir>\included.dat
File included using a directly downwards relative path to a file:	
<input_dir>\include_dir\included.dat	
File included using a non-directly downwards relative path to a file:	
<input_dir>\..\included.dat	<project_dir>\included.dat
<input_dir>\..\include_dir\included.dat	<project_dir>\included.dat
<input_dir>\dir\..\dir\included.dat	<project_dir>\included.dat

Component Selection

The components are the starting point in the Structural model window. Through the workflow for each component, the user is guided to

- select the region of the structural FEA model where Digimat material should be used. Several model sources are possible:
    - From experimental data: based on stress-strain curves, a Digimat material is automatically calibrated
    - From Digimat-MX: a Digimat material can be selected from the Digimat-MX database
    - From file: a Digimat material can be selected from the hard drive
- specify the source of manufacturing data, if needed (this step is skipped for CFRP process):
    - From simulation results: manufacturing data has been computed in a manufacturing process simulation tool (usually on a specific mesh), it will therefore need to be mapped to the structural mesh (see section Fields Mapping).
    - Estimation: fiber orientation is computed from fiber orientation estimation tool provided with Digimat-RP (see section Estimation).
    - From mapped files: manufacturing data is already available on the structural mesh. One or several files (see section Using Data from Multiple Files) containing the manufacturing data will be directly loaded.

  - User defined: a homogeneous microstructure based on the one specified in the Digimat material will be used.
- review the solution settings before submitting the FEA job.

When a structural model is loaded, components are identified for each material defined in the original structural FEA model (see Figure 3-3). Of course, not all components need to be updated with a Digimat material model. Only components corresponding to regions where a Digimat material needs to be added should be selected.

A component can be selected in two different ways:

- by selecting the relevant component in the component list
- by double-clicking on a region of the structural model (in the 3D viewport).

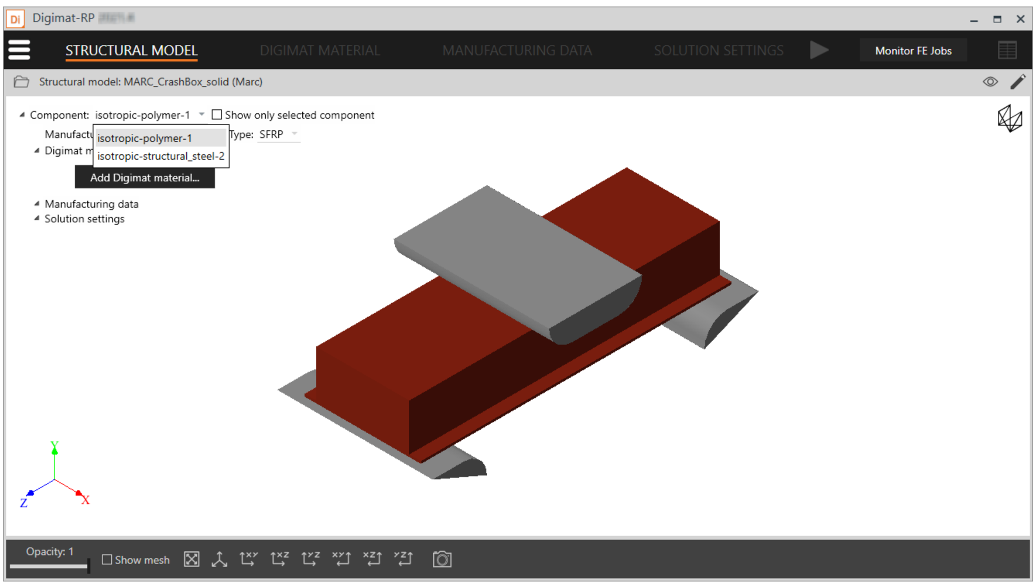

Figure 3-3 Components available for a newly loaded structural model.

If a component is not easily accessible in the 3D viewport, it is possible to visualize only the selected component by toggling on the **Show only selected component** option available besides the component list. Depending on the content of the input deck, some components may be restricted from the addition of a Digimat material if the resulting model or workflow is not supported by Digimat-RP. For example, this may happen if the corresponding original material was assigned to both solid and shell elements in the model, or if it was assigned to a layer of a non-homogeneous composite part/section.

Manual Workflow

In the legacy manual workflow where only the interface file is generated, the structural model window is simplified, as shown in Figure 3-4.

No structural model needs to be loaded, but the model format needs to be selected from the list of supported CAE codes. In most cases, the format can also be detected automatically by Digimat-RP after selecting the input deck file.

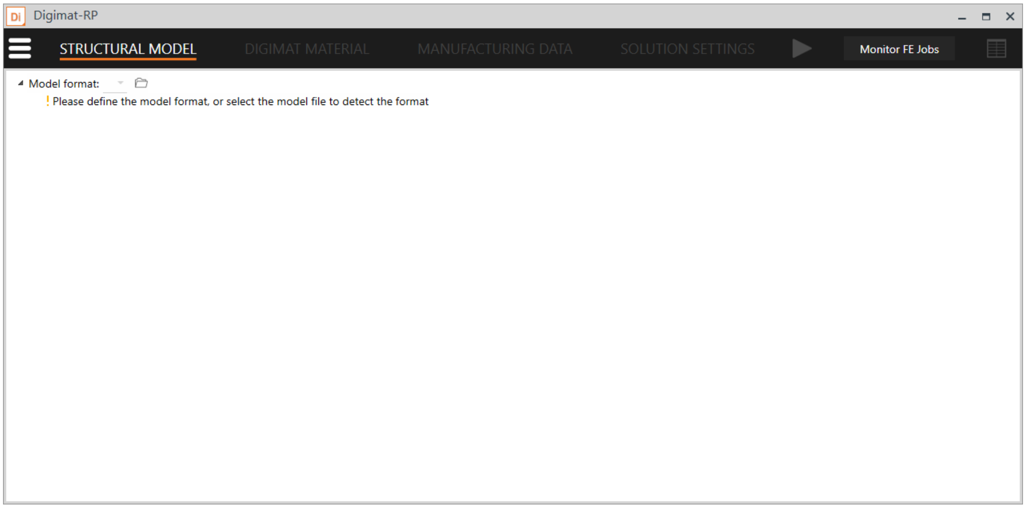

Figure 3-4 Simplified Structural model window at the beginning of the manual workflow.

Once the format is defined, the analysis type needs to be defined if the selected format supports both implicit analyses and explicit analyses. The remaining steps are similar to the standard **Structural Model**, except that no component selection is required since no model has been loaded (see Figure 3-5).

Figure 3-5 Simplified Structural model window after having selected the model format.

Structural Results

When combining CAEfatigue or nCode interface with the Run fatigue FEA from static FEA results workflow, the user does not need to load a structural model but a results file.

In such configuration, the **Structural model** window is called **Structural results**. The user has one single operation to do, load a results file. Once done, the user can proceed with the next step of the workflow which is to assign the material model.

The results file that can be loaded are only the one that are supported with CAEfatigue and/or nCode:

- `.odb`
- `.rst`
- `.t16`
- `.h5` or `.op2`

4 Material Window

- Model
- Solution Settings and Advanced Solver Settings

The **Digimat material** window contains the required features to define Digimat materials used in the coupled analysis. The window consists of one or several tabs, each one associated to a Digimat material that will be assigned to the current component. For example, CFF processes require the definition of two materials per component, as shown in Figure 4-1. Each material tab is itself divided into three other tabs: the **Model** tab which contains parameters related to the physical definition of the composite material, the **Solution settings** tab which contains the main parameters for the behavior of the material in the numerical analysis. A third tab contains all advanced solver settings which can be used to fine-tune the analysis. The Solution settings section can also be directly accessed via the **Solution settings** tab in the ribbon, or by clicking on **Review...** at the end of the component workflow (see section Solution Settings for more information).

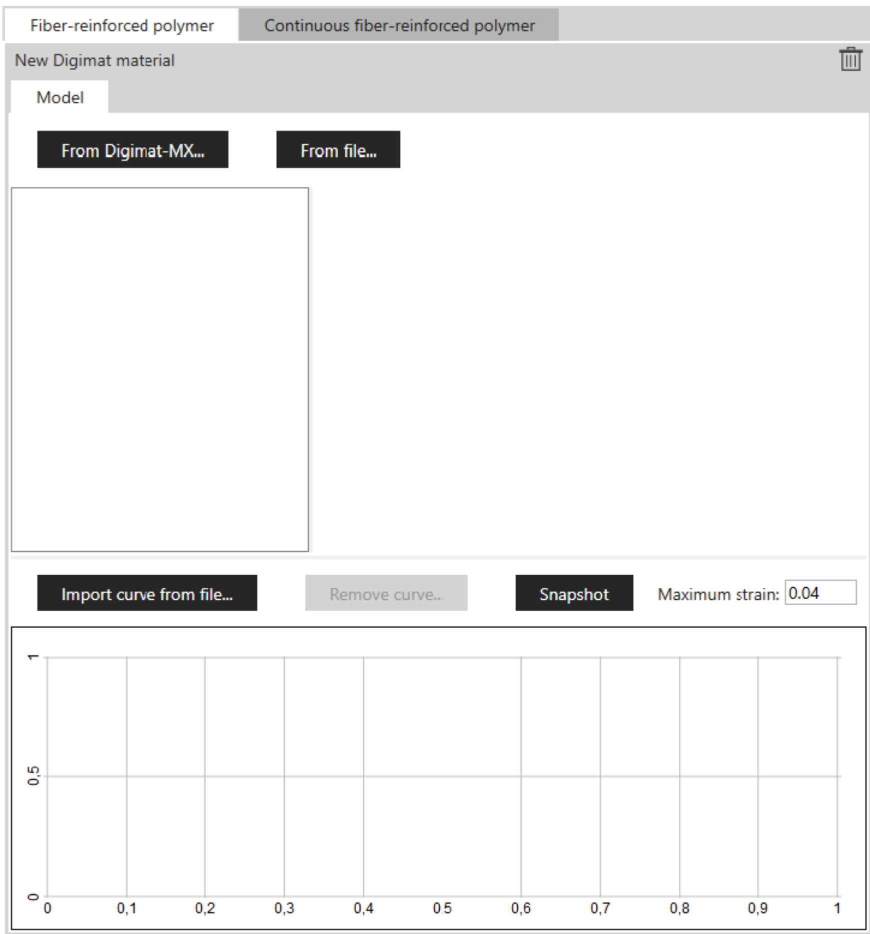

Figure 4-1 The Digimat material window allows to define the required Digimat materials. Here, two materials need to be defined: a fiber-reinforced polymer, and a continuous fiber-reinforced polymer.

Model

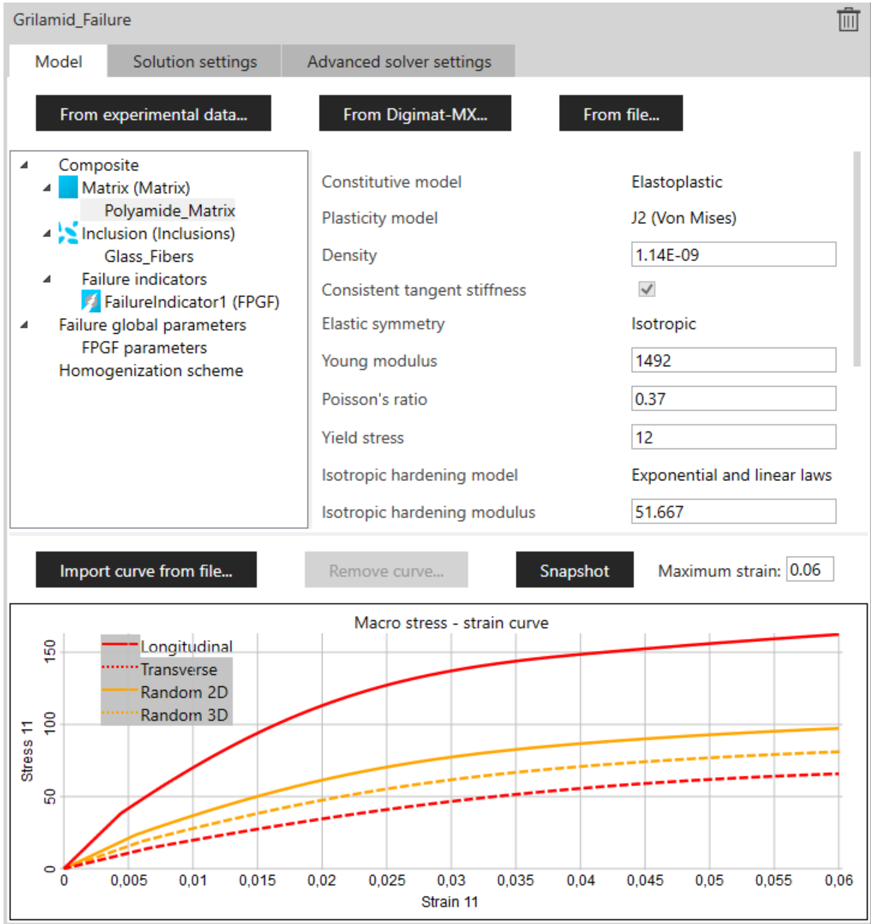

Figure 4-2 The Model tab.

The Model tab of the Digimat material window is in turn divided into two parts (see Figure 4-2).

The upper part contains a tree representation of the composite material, each item of the tree being detailed upon selection.

The lower part contains a stress-strain plot (uniaxial tension test) for the material for different types of microstructure or loading conditions.

Material Model Sources

Depending on the type of manufacturing and material defined for the component, different options are available to specify the Digimat material to use.

From Experimental Data

This option (available for SFRP or LFRP materials with injection or compression molding manufacturing process) allows creating a Digimat material model based on reverse engineering of experimental data. The creation of the material model first involves a definition step, where input information is required:

- description of grade: matrix material, fiber material and fiber amount,
- selection of required performance: linear stiffness, non-linear stiffness or non-linear stiffness combined with failure,
- input of experimental data: stress-strain curves from at least two loading directions, which can imported either by copy-paste or by text file selection,
- definition of the microstructure source: fiber orientation description used for the reverse engineering can be either identified automatically, or user-defined via the import of a `.csv` file following the CSV Laminate File format.

Once definition is complete, reverse engineering can be executed by clicking on Proceed. Reverse engineering computation will then start based on the number of CPUs defined. After completion of the reverse engineering, a new Results tab becomes available which allows to review the obtained parameters as well as the fit between experimental data and Digimat model response for the corresponding loading angles. The obtained material model can be exported as a `.daf` file by selecting the **Export model...** button available in the results tab.

From Digimat-MX

The material definition can also be obtained from the Digimat-MX database. Selecting this option will open up Digimat-MX and will pause Digimat-RP. It is then possible to browse through the Digimat-MX material database to select the desired material. Once a material definition has been selected, closing Digimat-MX will trigger the import of the material in Digimat-RP.

From File

This option allows loading an existing material definition from a `.daf/.mat` file. If the selected material model already contains a link to a manufacturing data file for some microstructure parameters (e.g., fiber orientation), the usage of a manufacturing file is strictly required to setup the coupled analysis.

Unit System

When loading a new material model, the parameters are to be converted to the unit system defined for the structural model. The conversion is done automatically for materials imported from a Digimat-MX database. When loading data from a `.dat/.mat` file, the conversion is automatically done if the material file contains a unit system definition, otherwise a pop-up window is shown, asking for the unit system used in this material file (see Figure 4-3).

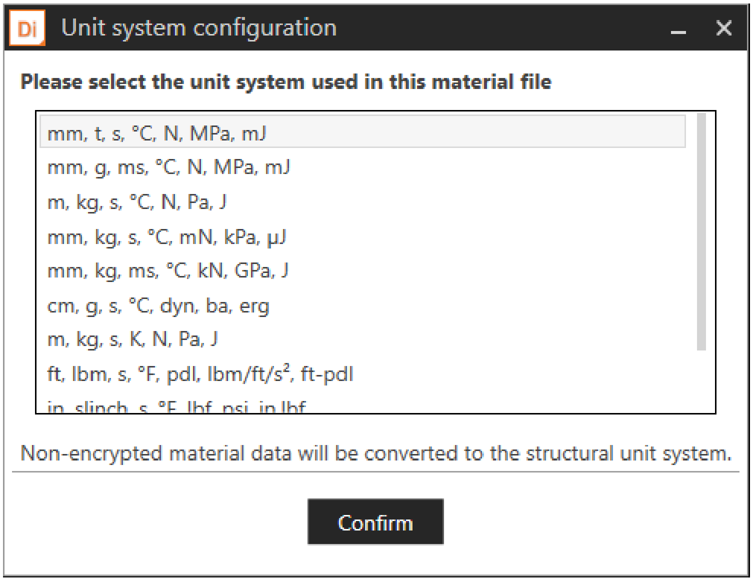

Figure 4-3 Selecting the unit system to use in Digimat-RP.

Editing the Material Definition

Some parameters of the material definition can be edited. Each time a parameter is modified, the macro stress-strain curve is updated to illustrate the effect of the modification (except if the modification makes it invalid). For each editable parameter, continuous validation is used to ensure that the material definition is valid. Each invalid parameter is highlighted with a red cross and shows a tooltip indicating the reason of the error. The invalid items in the tree will also present the same icon.

Fatigue Failure Indicators

When working with implicit FE models, fatigue failure indicators are available by loading a material file that already contains a fatigue failure indicator (see section High Cycle Fatigue). In that case, S-N curves are displayed instead of the macro stress-strain curves (see Figure 4-5). Note that it is not allowed to combine fatigue failure indicators and standard failure indicators.

Figure 4-4 Invalid parameters are highlighted.

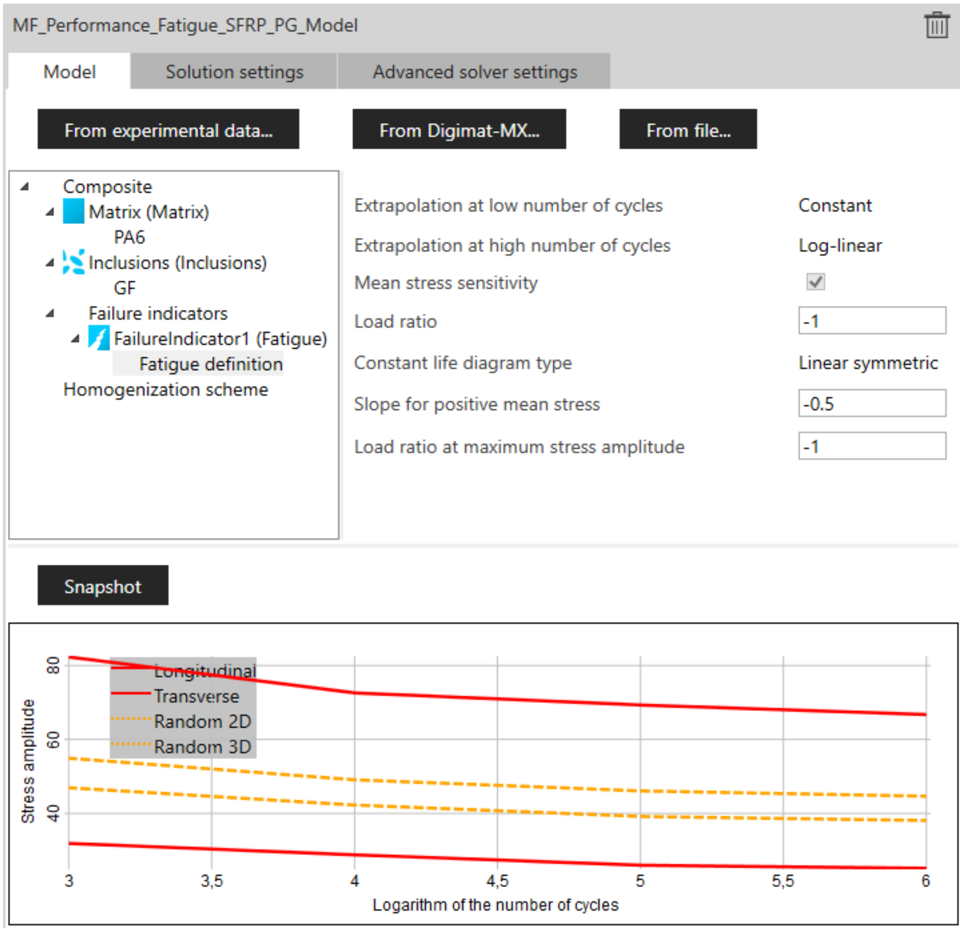

Figure 4-5 S-N curves are displayed when the selected Digimat material contains a fatigue failure indicator.

Solution Settings and Advanced Solver Settings

Please refer to section Solution Settings for more information.

5 Manufacturing Data Window

- Injection and Compression Molding
- Fused Filament Fabrication
- Fused Deposition Modeling
- Continuous Fiber Fabrication
- Selective Laser Sintering
- CFRP Process
- Fields Mapping

The manufacturing data window is accessible as soon as the Digimat materials required by a component have been defined and are valid.

It can be accessed from the Manufacturing data tab in the top ribbon or via the **Define manufacturing data** button in the component workflow.

The manufacturing data step aims at defining the relevant manufacturing information for the chosen component. In a general way the manufacturing data possibilities vary as a function of the application:

- injection molding and compression molding
  - SFRP: fiber orientation, weld line, weld surfaces and residual stresses
  - LFRP: fiber orientation, fiber volume fraction, fiber length, weld line, weld surfaces and residual stresses
- fused filament fabrication: toolpath (G-code and Markforged) and residual stresses
- fused deposition modeling: toolpath (Insight) and residual stresses
- selective laser sintering: global printing direction and residual stresses
- continuous fiber fabrication: toolpath (G-code and Markforged) and residual stresses for the fiber reinforced polymer type

For a given application, there are several possibilities to define the manufacturing data. The next sections explore these possibilities in more details.

When combining CAEfatigue or nCode interface with the Run fatigue FEA from static FEA results workflow, the user only has access to the option **From mapped files** since there is a guarantee that the user already performed the mapping to run the static analysis.

When using the legacy manual workflow to manually generate the interface file, the user only has access to the two options **From mapped files** and **User-defined**. This is due to the fact that no structural model is actually loaded in Digimat-RP so the mapping also has to be managed manually outside Digimat-RP.

In the particular case of SLS, there is actually no required data to map but printing parameters have to be defined at the process level. Hence, in the manual workflow, only **From simulation results** is available for SLS materials and initial stresses are not supported in this case in Digimat-RP.

Injection and Compression Molding

Injection and compression molding related features and options in the manufacturing data window are discussed in this section.

From Simulation Results

When selecting **From simulation results**, the manufacturing data window is made of two tabs:

- **Data selection** tab: enables to specify and load manufacturing simulation mesh and the corresponding manufacturing data. The required fields are marked with a red "Required" mark and cannot be untoggled from the selection.

- **Mesh visualization** tab: enables to visualize the manufacturing data. If several manufacturing data have been loaded, field selection is available at the bottom of the Manufacturing data window via the **Plot** drop-down menu.

This option is not supported for the continuous fiber-reinforced polymer type of continuous fiber fabrication.

The following mesh file formats are supported:

- Marc (`*.dat`)
- MSC Nastran (`*.bdf, *.dat, *.nas`)
- Abaqus (`*.inp`)
- ANSYS (`*.cdb, *.inp, *.dat, *.ans`)
- LS-DYNA (`*.k, *.key, *.dyn`)
- OptiStruct (`*.fem`)
- PAM-CRASH (`*.pc`)
- PERMAS (`*.uci`)
- Samcef (`*.dat`)
- Patran (`*.pat`)
- Moldflow (`*.udm`)
- 3D TIMON (`*.msh`)
- RADIOSS (`*.rad`)
- Cadmould 3D-F (`*.cfe`)
- STL geometry (`*.stl`)

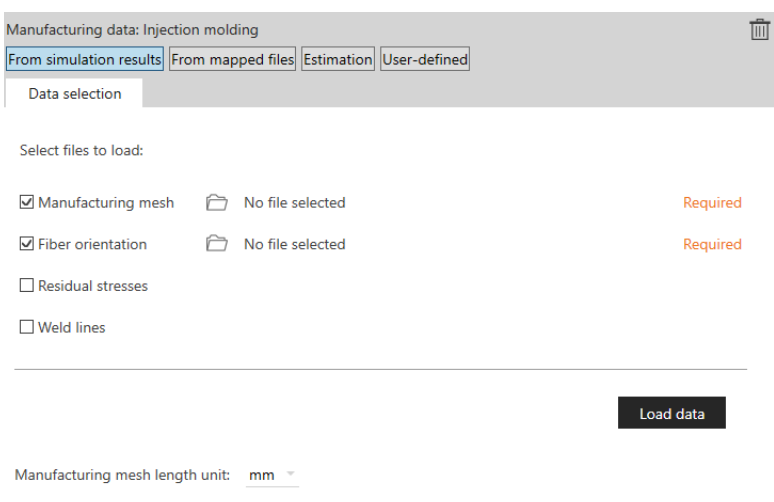

Figure 5-1 Selecting the manufacturing data to use in Digimat-RP.

Manufacturing data from the following simulation codes are supported:

- Moldflow 3D (`*.xml`): fiber orientation, weld line (angles) and weld surfaces, fiber length, fiber volume fraction, residual stresses
- Moldex 3D (`*.o2d, *.nwd, *.fld, *.fcd, *.s2d`): fiber orientation, weld line (angle), fiber length, fiber volume fraction, residual stresses
- Sigmasoft (`*.xml`): fiber orientation, weld line (criterion), residual stresses
- Simpoe (`*.xml`): fiber orientation, residual stresses
- 3D TIMON (`*.bou, *.fou, *.wldang, *.unv, *.flunv, *.vfunv`): fiber orientation, weld line (angle), fiber length, fiber volume fraction
- 3D TIMON Light3D (`*.unv`): fiber orientation
- Rem3D (`*.mtc`): fiber orientation
- Moldflow midplane (`*.ele.001, *.xml, *.nod`): fiber orientation, weld line (angle), fiber length, residual stresses
- Moldex 3D midplane (`*.ele.001, *.nwd`): fiber orientation, weld line (angle)
- 3D TIMON midplane (`*.dat, *.fou, *.wldang`): fiber orientation, weld line (angle)
- Cadmould 3D-F (`*.txt, *.car`): fiber orientation
- VISI Flow (`*.xml`): fiber orientation

Besides, data files in Digimat format are also supported (`*.dof`): fiber orientation, weld line, fiber length, fiber volume fraction, residual stresses

Once manufacturing data has been loaded in Digimat-RP a mapping step is required to transfer it to the structural model. The mapping can be launched by clicking the **Map fields...** button in the component workflow tree. For more details on the mapping procedure, please refer to section Fields Mapping. For SMC material models, only orientation and weldline data are supported.

It is not required for manufacturing data to be available on the whole manufacturing, provided that the whole associated region of the structural model has available data after the mapping operation.

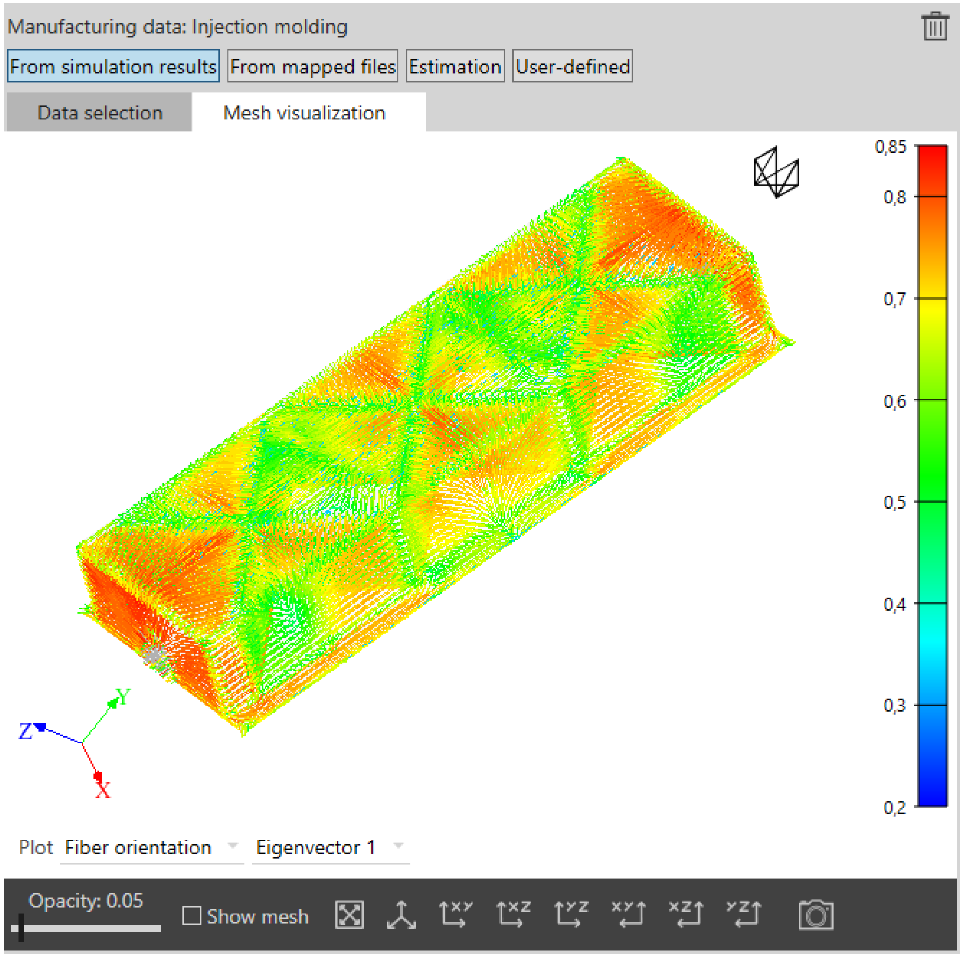

Figure 5-2 Visualization of manufacturing data.

From Mapped Files

If a manufacturing data file already corresponding to the structural model is available, it is possible to load it directly by choosing the **From mapped files** option. Similarly to the **From simulation results** option, fiber orientation data is mandatory, and additional data types are also supported. Please refer to From Simulation Results for the detailed list of supported data. Note that some formats may be unavailable when using **From mapped files**, e.g., if a mapping is always required to use the file data.

Once path to the mapped files is specified, loading of the data to the structural model is achieved via the **Load data** button.

For SMC material models, only orientation and weldline data are supported.

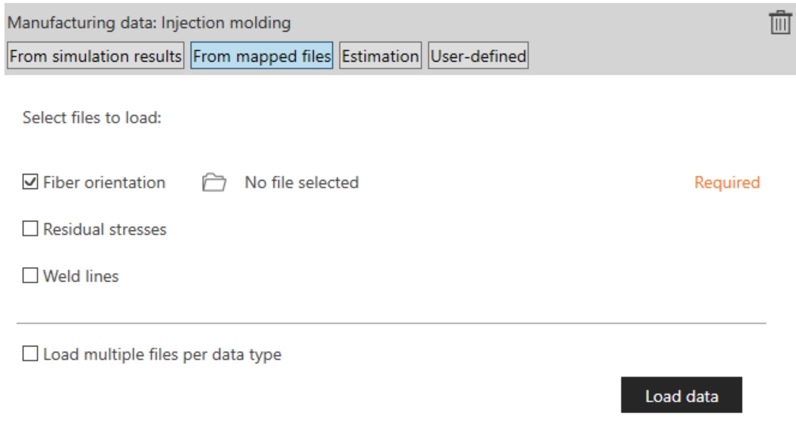

Figure 5-3 Selection mapped files for manufacturing data.

Using Data from Multiple Files

When using manufacturing data files containing data already available on the structural mesh, multiple files may be selected by activating the **Load multiple files per data type**. In that case, another window will be displayed, showing selected files with a summary of the available data (see Figure 5-4). In order to proceed, data must be available on all elements of the associated region. Note that when an element has data available in several files, only the first one will be taken into account. Files may be deselected by toggling the associated checkbox, and may be removed from the list by selecting them and pressing the **Delete** key.

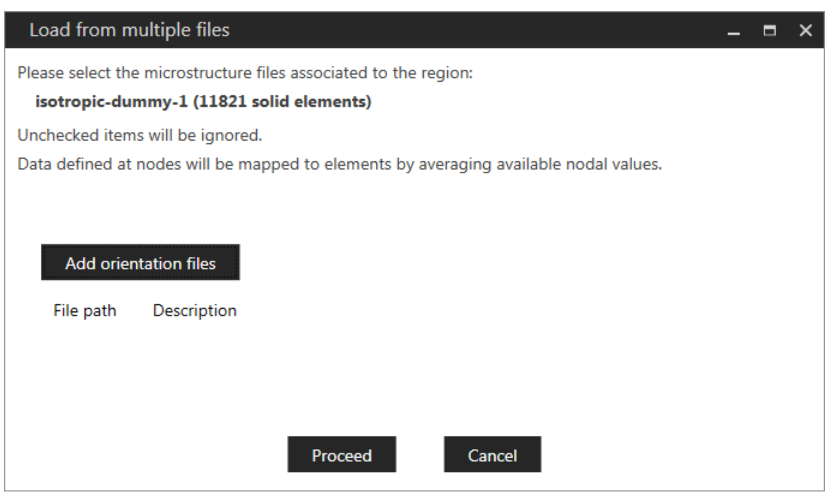

Figure 5-4 Importing orientation data from several files on a single region of the structural model

Estimation

Overview

The estimation is only supported for Injection molding manufacturing process.

Accurate analysis of reinforced plastic parts usually requires access to microstructure information such as fiber orientation. Such information can be available from most injection simulation software, and loaded in Digimat-RP for coupled analysis setup. However, access to such data is sometimes cumbersome due to the long time required for a complete injection simulation, or the fact that injection simulations are run by other departments inside a company.

To speed up the analysis of reinforced plastic part, a microstructure estimator is accessible within Digimat-RP. It enables to easily and efficiently estimate fiber orientation (for both 2D and 3D meshes) and weld line location (only for 3D meshes) within a part, based on very limited user input.

The microstructure estimator is powered by Moldex3D technology with specific user guidance for Digimat-RP for the unique purpose of estimating fiber orientation and weld line location.

Below is a description of the workflows available to the user for microstructure estimation, as well as the associated main capabilities.

Microstructure Estimation Process

Upon selection of the manufacturing data source, the user can select “Estimation”, giving the choice to two workflows, either “From structural mesh” or “From custom model” (see Figure 5-5). The user is then prompted to browse and select a mesh file, as typically supported by Digimat-RP. For more details on the possible workflows and their application, please refer to the next section dedicated to this topic.

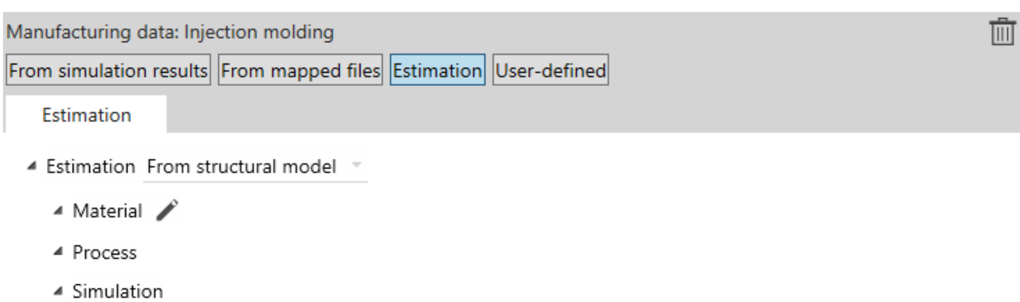

Figure 5-5 Selection of microstructure estimation as source of microstructure.

Microstructure estimator setup: Once the mesh is selected, the microstructure estimator can be set up.

Three steps are required in the definition:

- material,
- process,
- simulation.

Material selection is done in the database where grades are sorted by type of resin. Upon selection, the rheological model required for the filling analysis is automatically selected. Please note that there is no restriction on the selection of a material in the fiber orientation estimator based on the material selected for the structural model.

Once material is selected, some process settings are automatically suggested:

- melt temperature,
- mold temperature,
- filling time.

These settings are computed or extracted from Moldex3D, and it is generally not necessary to modify them. If the user has some deeper knowledge and details about process settings values, it is possible to adapt them (see Figure 5-6). Please note that for large parts, suggested filling time can be high, and may require user adjustment towards lower values.

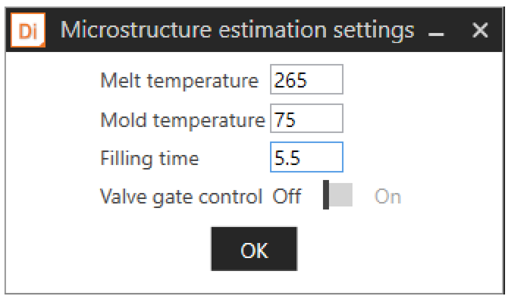

Figure 5-6 Process settings: melt and mold temperatures, filling time.

The last step in the process part is the definition of injection gates. Once a gate definition box is open — which is possible by clicking on **Add injection point** — the user can pick the gate position by double-clicking on the model visualization or by directly entering X, Y, Z coordinates after clicking on **Edit coordinates**. When defining a position through coordinates, the closest surface node of the mesh will be used as actual injection point, and the displayed coordinates will be updated. The gate diameter is set by default to 1 millimeter, and can be adapted if needed. Once both the gate position and the diameter are set, the user can confirm his input by clicking on **OK**. Additional gates can then be defined. Any gate definition can be edited by clicking on the corresponding **Edit** icon in the injection point table. Once all inputs are set in the microstructure estimator, the user can proceed to simulation run by clicking on **Run...**

Definition of valve gate controls: By default, when simulating an injection process with multiple gates, all gates are opened at the same time at the start of the injection, and closed at the same time following the filling time defined in the process settings. Valve gate controls can be used to define the opening and closing times of each gate, individually. This feature is a key component of multi-gates systems used to avoid issues such as weld lines. Once injection gates have been defined, valve gate settings can be modified in the process settings (see Figure 5-8). Each gate can only be opened and closed once during the injection process. The total filling time is automatically set to the latest valve gate closing time.

Microstructure estimator run To run the microstructure estimation, the user is required to specify the number of CPUs used for injection simulation run as well as the working directory. A maximum of 4 CPUs is possible. Microstructure estimator happens in three fully automatic steps:

- injection model creation including automatic meshing,
- filling analysis and fiber orientation computation,
- transfer of the computed fields onto the mesh specified in the Fiber orientation estimator.

Note:	The mesh loaded in Digimat-RP should be a proper mesh in order to successfully simulate the injection process. For example, the model should not contain free edges, overlapping elements, extremely distorted elements, etc.

Typical run time on 4 CPUs and RAM consumption range within 10min — 1h and 1 – 3 GB. Additional RAM will be consumed during transfer of the computed fields onto the mesh selected in Digimat-RP.

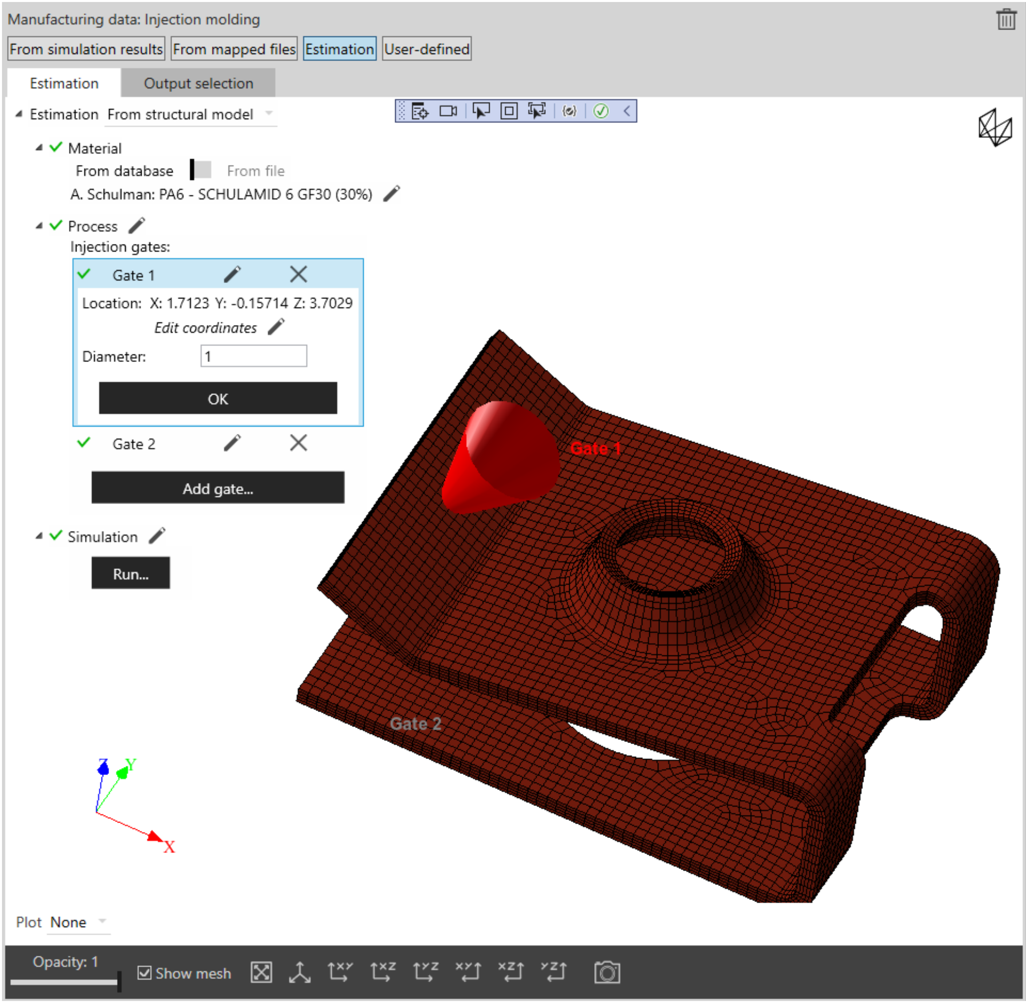

Figure 5-7 Setup of microstructure estimation.

Additionally, it is possible to access to fiber orientation prediction parameters, via the **Edit settings** button in the Simulation step. These settings can influence the fiber orientation prediction, they are restricted to advanced usage of the software. Several rotary diffusion models are available, for both short and long fiber reinforced plastics:

- SFRP: Folger-Tucker and iARD
- LFRP: ARD and iARD

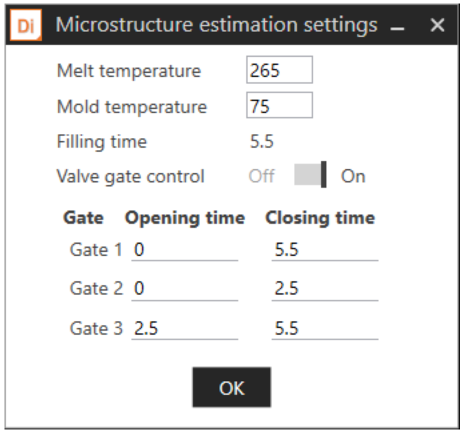

Figure 5-8 Valve gate settings. Each gate has its own filling time interval.

Post-processing: Once the transfer of injection results onto the initial mesh used for microstructure estimation has been completed, the user can post-process them inside Digimat-RP. Available results consist of:

- fiber orientation tensor components,
- fiber orientation vector plots,
- melt front time and melt front time animation,
- weld lines. (available when running the estimation on a 3D mesh only).

Melt front time is accessible by selecting the **Melt front time** field in the Plot drop-down menu. By default, a static view of the melt front time is plotted (see Figure 5-9). An animation can be created by toggling **Show animation** and clicking on **play**. The animation can also be saved as a `*.gif` file. For best melt front animation visualization, it is recommended to set the opacity to a low value.

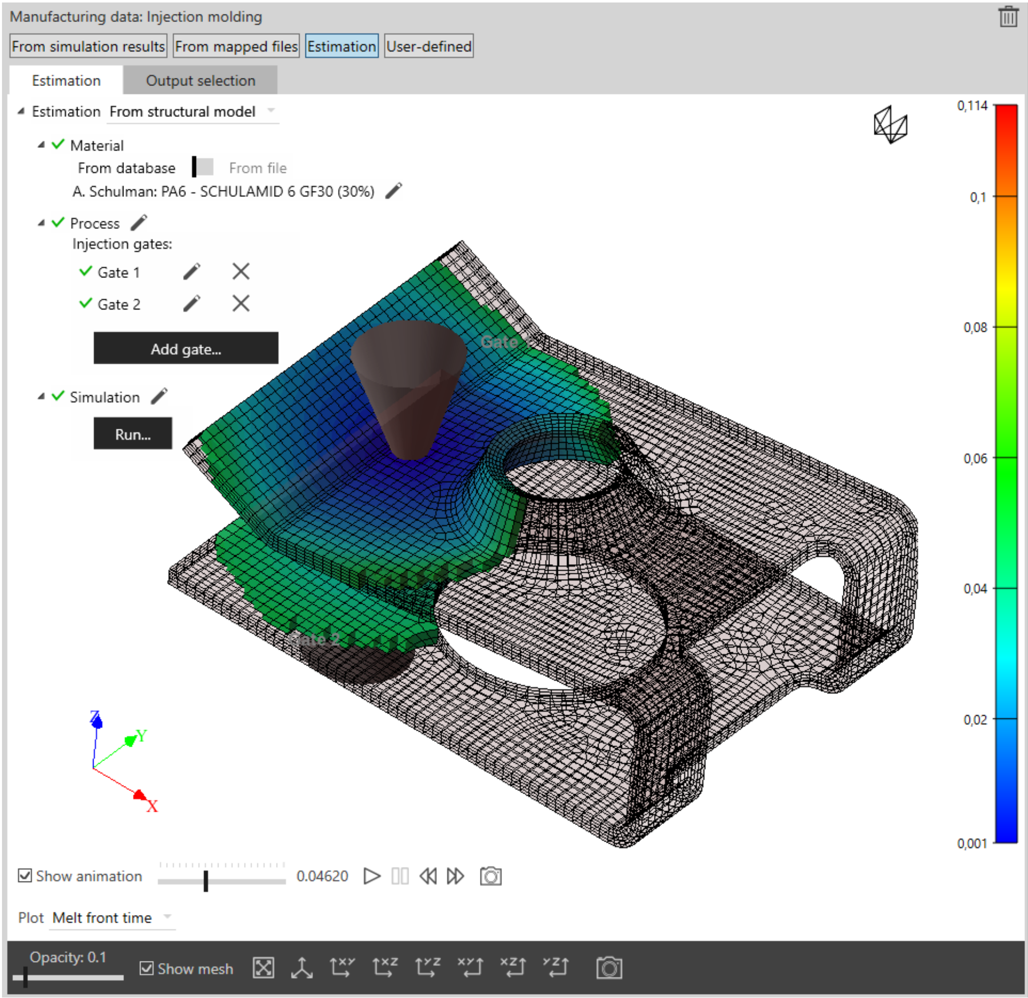

Figure 5-9 Visualization of melt front animation.

Once injection results have been obtained, the tab **Output selection** appears, allowing to select results to use in the structural model (see Figure 5-10). When using the "From structural mesh" workflow, weld line radius and angle thresholds can be set here, and the Fiber orientation file can also be exported by clicking on **Export orientation file**.

Workflows

As already introduced, two workflows for fiber orientation estimation are available inside Digimat-RP, namely:

- From structural model,
- From custom model.

From structural model will allow to work on the component selected in the structural model. This workflow is the recommended one for parts where injected part corresponds to the structural model. **From custom model** allows to open the microstructure estimator using any user-specified mesh, giving extra flexibility. This flexibility is particularly required for two scenarios: using sub-modeling for structural analysis or using a shell structural model in combination with a 3D solid mesh for fiber orientation estimation. When sub-modeling in used in a structural application, the structural mesh does not correspond to the mold design anymore and thus cannot be used as a basis for injection simulation.

Usage recommendations and limitations

Microstructure estimation comes with the following limitations:

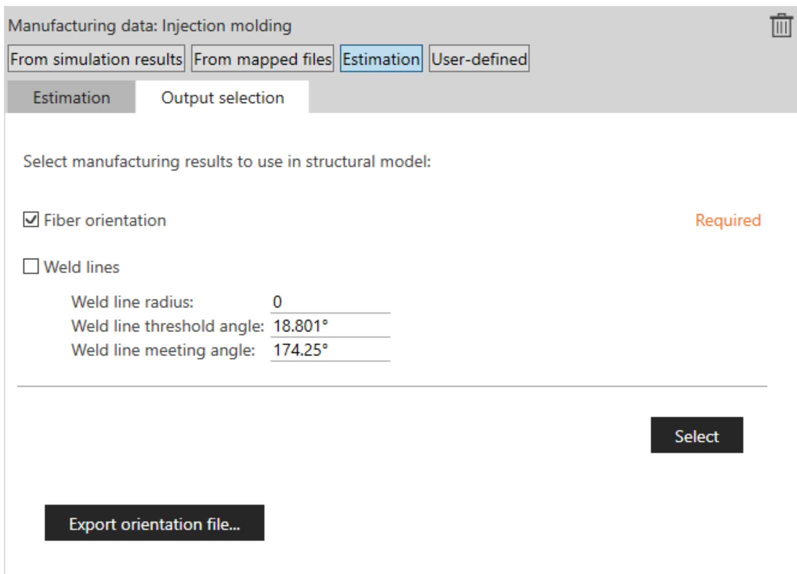

Figure 5-10 Output selection after running injection simulation.

- The mesh loaded for microstructure estimation cannot contain any element described with a negative Jacobian. Such meshes can lead to instabilities of the software and are not supported.
- Parallel computation with Moldex3D uses Intel-MPI. It is recommended to read the installation section to avoid any conflict with other Intel-MPI services (see section Installation of Intel-MPI 2019 Hydra Service for Fiber Orientation Estimation *in the Digimat Installation and Operations Guide*).

For best usage of the microstructure estimator, it is recommended to apply the following best practices:

- Short shot can happen with improper process settings definition. A short shot consists of an incomplete injection, due for instance to the melt freezing inside the mold. If a short shot happens, user will be notified in the monitoring window. The setup of a coupled analysis will then be blocked, since no microstructure is available for the full structural mesh. To prevent short shots, it is recommended to keep a low time as well as the default process conditions. Gate definitions may need to be revised for larger parts.

- Gate definitions must be such that no intersection happens between gates. If intersecting gates are defined, an error will appear when creating the injection simulation model, making the microstructure estimator run unsuccessful.
- RAM consumption can become significant when working with input meshes for microstructure estimation which contain more than 200k elements. It is recommended to have a minimum of 16 GB of RAM for smooth usage of the fiber orientation estimator.

User-defined

When manufacturing data is hardly available from simulation, mapped files or estimation, basic user-defined microstructures can be defined through the **User-defined** option. This option enables to define homogeneous microstructure information for the full component, e.g., a constant fiber orientation tensor, a constant fiber aspect ratio... These values can be defined from the Digimat material tab, as indicated by the Manufacturing data window when selecting the **User-defined** option. The **User-defined** option for Manufacturing data is only available when the Digimat material model used in the component comes from Digimat-MF, i.e., without an existing definition of a manufacturing data file for a microstructure parameter (e.g., fiber orientation file to describe the fiber orientation).

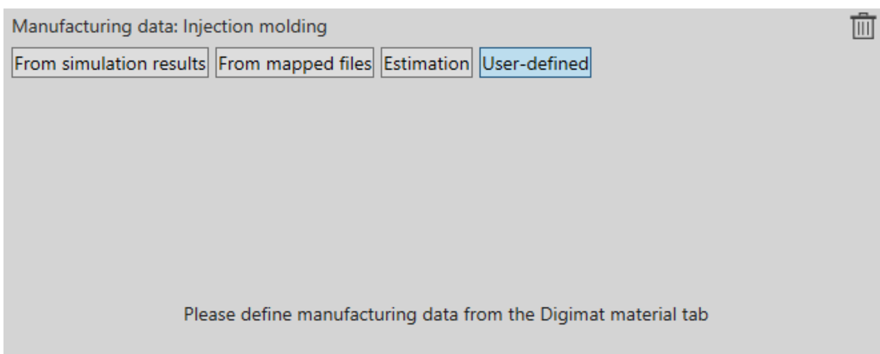

Figure 5-11 Selecting user-defined manufacturing data in Digimat-RP.

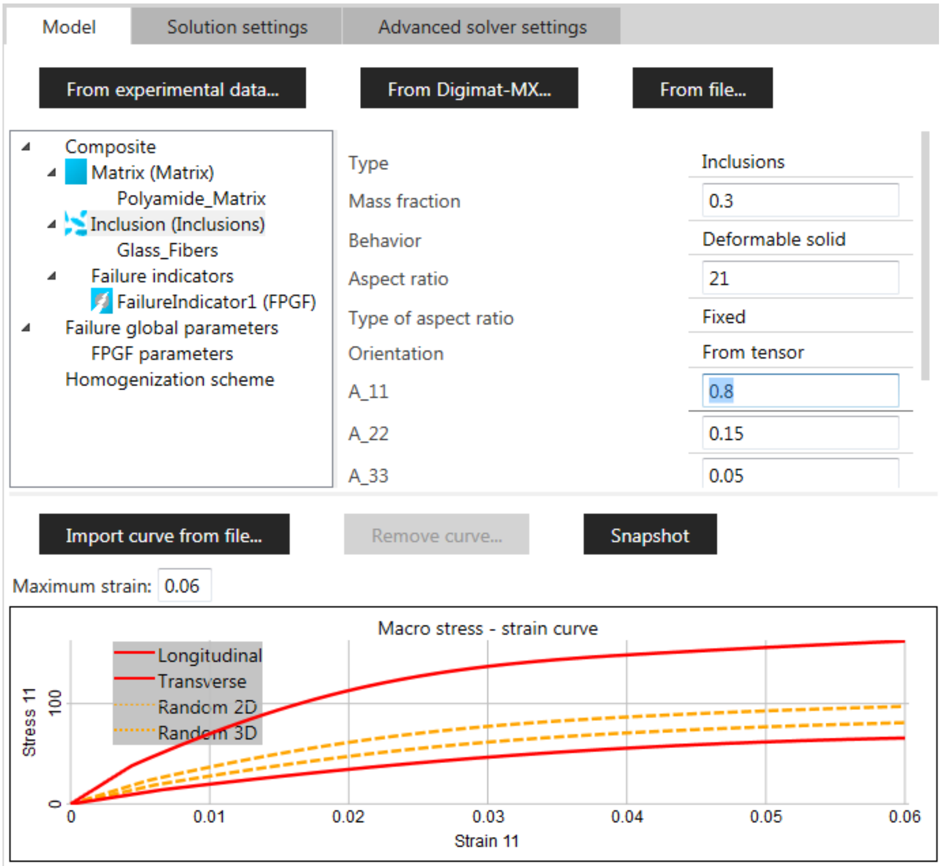

Figure 5-12 Defining homogeneous microstructure in Digimat-RP.

Density Mapping From the Volume Fraction Data

A volume fraction distribution may be taken into account to adapt the density at the element level if these conditions are met:

- fiber volume fraction data in Digimat format are loaded (LFRP only),
- the CAE code is MSC Nastran/SOL400 (NVH),
- the loaded structural model correspond to an NVH analysis.

This is then done by checking the option Density discretization under NVH controls in the Solution settings tab. To this end, the range of volume fraction is uniformly discretized with a user-defined number of intervals (see Figure 5-13).

Figure 5-13 Controlling the density definition as a function of the fiber volume fraction in Digimat-RP.

Fused Filament Fabrication

When working in a fused filament fabrication (FFF) application, two options are available: **From simulation results** and **From mapped files**. For fiber reinforced polymers, the fiber orientation has to be specified in the Digimat material by modifying the orientation of the inclusions. The required manufacturing data for FFF is toolpath information (which can be exported from a slicing software) defined by:

- Tessellation geometry file (`.stl` or `.3mf` file)
- Toolpath file (usually `*.gcode` or `*.mft`)

Optionally, residual stresses obtained from Digimat-AM may be imported, requiring two files:

- Digimat-AM mesh (`*.dat`)
- Residual stresses from Digimat-AM (`*.xml`)

The below part describes the workflow for the option **From simulation results**. When working with both toolpath and residual stresses data it is recommended to use tessellation (STL or 3MF) and `.dat` files which correspond to the same model, i.e., with an exact superposition.

The manufacturing data window is made of two to three tabs:

- Data selection tab: enables to specify and load manufacturing simulation mesh and the corresponding manufacturing data. The required fields are marked with a red **Required** mark and cannot be untoggled from the selection.
- Geometry visualization: enables to visualize the toolpath information.
- Mesh visualization tab: enables to visualize the residual stress data.

Once manufacturing data has been loaded, a mapping step is required, as described in Fields Mapping. When the mapping step is complete, the next step in the component workflow becomes accessible: Review Solution settings.

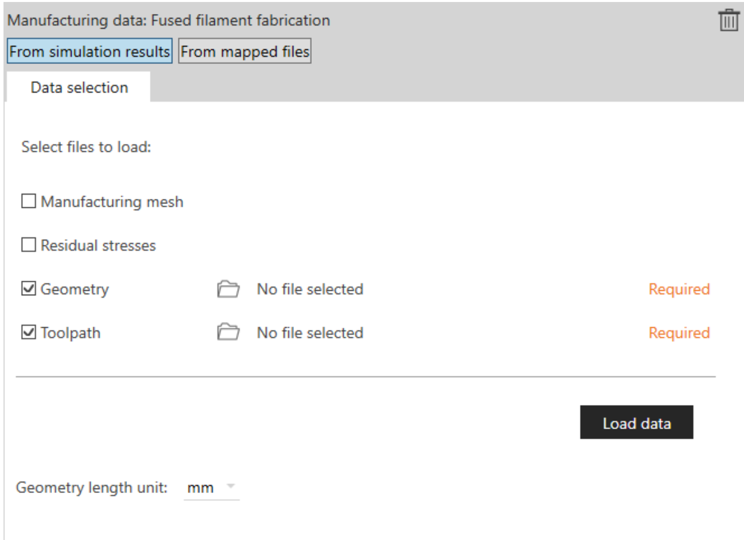

Figure 5-14 Selecting FFF manufacturing data in Digimat-RP.

Fused Deposition Modeling

When working in a fused deposition modeling (FDM) application, two options are available: **From simulation results** and **From mapped files**. For fiber reinforced polymers, the fiber orientation has to be specified in the Digimat material by modifying the orientation of the inclusions. The required manufacturing data for FDM is toolpath information, defined by:

- Tessellation geometry file (`.stl` or `.3mf` file)
- Toolpath file (`*.txt`): this file can be exported from the Insight software, starting from release 11.8

Optionally, residual stresses obtained from Digimat-AM may be imported, requiring two files:

- Digimat-AM mesh (`*.dat`)
- Residual stresses from Digimat-AM (`*.xml`)

When working with both toolpath and residual stresses data it is recommended to use tessellation (STL or 3MF) and `.dat` files which correspond to the same model, i.e., with an exact superposition.

The manufacturing data window is made of two to three tabs:

- **Data selection**: enables to specify and load manufacturing simulation mesh and the corresponding manufacturing data. The required fields are marked with a red **Required** mark and cannot be untoggled from the selection.
- **Geometry visualization**: enables to visualize the toolpath information.
- **Mesh visualization**: enables to visualize the residual stress data.

Once manufacturing data has been loaded, a mapping step is required, as described in Fields Mapping. When the mapping step is complete, the next step in the component workflow becomes accessible: Review Solution settings.

Continuous Fiber Fabrication

When working in a continuous fabrication (CFF) application, two options are available: **From simulation results** and **From mapped files**. Similarly to FFF and FDM, the fiber orientation has to be specified in the Digimat material by modifying the orientation of the inclusions. The required manufacturing data for CFF is toolpath information, defined by:

- Tessellation geometry file (`.stl` or `.3mf` file)
- Toolpath file (`*.mft`): this file can be exported from the Eiger software

Optionally, residual stresses obtained from Digimat-AM may be imported, requiring two files:

- Digimat-AM mesh (`*.dat`)
- Residual stresses from Digimat-AM (`*.xml`)

When working with both toolpath and residual stresses data it is recommended to use tessellation (STL or 3MF) and `.dat` files which correspond to the same model, i.e., with an exact superposition.

The manufacturing data window is made of two to three tabs:

- **Data selection**: enables to specify and load manufacturing simulation mesh and the corresponding manufacturing data. The required fields are marked with a red **Required** mark and cannot be untoggled from the selection.
- **Geometry visualization**: enables to visualize the toolpath information.
- **Mesh visualization**: enables to visualize the residual stress data.

Once manufacturing data has been loaded, a mapping step is required, as described in Fields Mapping. The mapping step will compute orientations but also material assignments based on the toolpath. Several element sets will be displayed on the original model (for example, an element set for all elements containing the plastic material but not the continuous fiber material, as printed).

When using the **From mapped files** option, both the filament orientation file and the material assignment file need to be loaded before progressing.

Once the material assignment data is available, the last step is to determine to which elements each one of the two Digimat materials needs to be assigned. This is done by selecting an element set and a Digimat material in the **Data selection** tab. The remaining Digimat material will be automatically assigned to the unselected elements. A standard recommendation is to assign the continuous fiber-reinforced Digimat material to the elements containing the fiber material.

Selective Laser Sintering

When working in a selective laser sintering (SLS) application, the only available option is **From simulation results**. The required manufacturing data for SLS depends on the material type:

- for **Unfilled** and **Bead-reinforced polymer** material, the only required input is the global printing direction, defining the Z direction used in the printing environment (see Figure 5-15). This printing direction determines the anisotropy axis used for the material failure indicator.

  The global printing direction is defined by two angles (θ and ϕ), which are similar to the angles defining a mechanical loading in Digimat-MF, the only difference being that the 1-2-3 axis are replaced by the X-Y-Z global axis of the structural model. When defining the θ and ϕ angles in Digimat-RP, it is possible to analyze the definition of the printing direction through the components of the vector, describing the components of the printing direction in the structural model axis system.

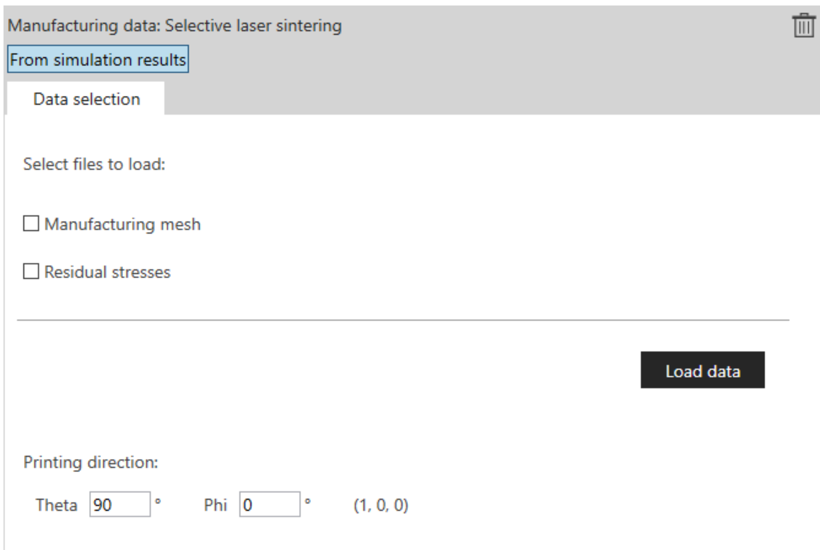

Figure 5-15 Defining SLS manufacturing data in Digimat-RP.

- for **Fiber-reinforced polymer** material, the following inputs are required (see Figure 5-16):

  – the **building** and **recoater** directions that define the printer axis system, expressed in the model axis system;

  – three angles defining the rotations applied to the model, in the printer axis system; these rotations are applied in Z-Y-X order, with intrinsic definition (rotation around the **updated** axes). A visualization of a sample part (a dumbbell aligned in the original X direction) is shown to confirm the orientation defined by the three angles.

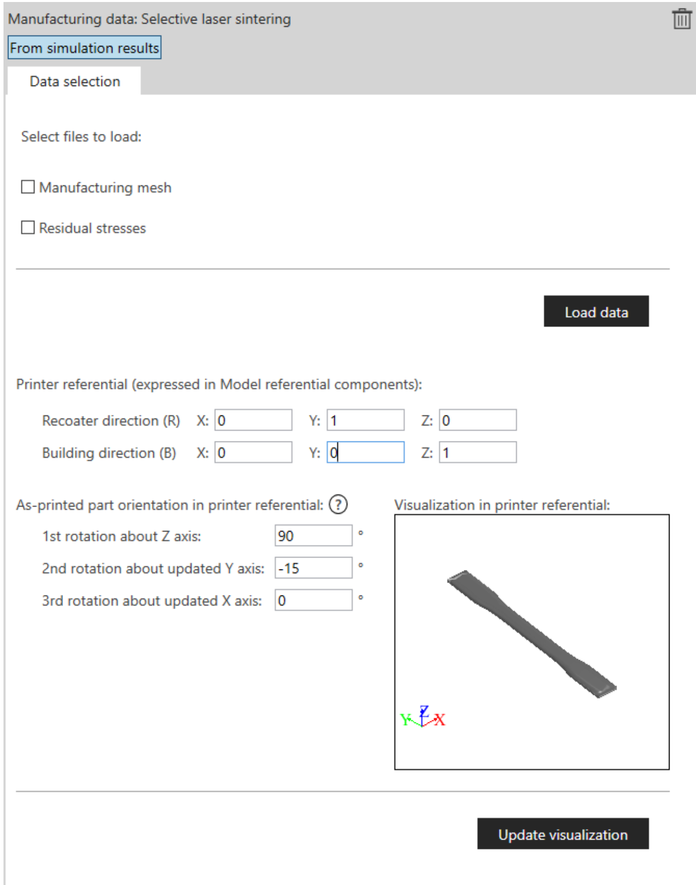

Figure 5-16 Defining SLS manufacturing data in Digimat-RP (for "Fiber-reinforced polymer" material).

Those inputs determine the anisotropy axis used for the material failure indicator, but also the final orientation tensor used for the material in the model axis system. This is done by performing a rotation of the original fiber orientation (given loaded the initial material file), from the printer coordinate system to the model axis system.

Remarks:

– The fiber orientation must be defined as an orientation tensor for the material to be valid for this workflow.

– After loading the material file, this orientation tensor cannot be modified.

To ease up understanding of what will be the final orientation tensor, the printer axes are displayed (and automatically updated) in the model plot (see Figure 5-17). The final orientation tensor is also displayed in the model plot; however, the user has to click on the **Update visualization** button for the model plot to reflect the changes made in the manufacturing data definition.

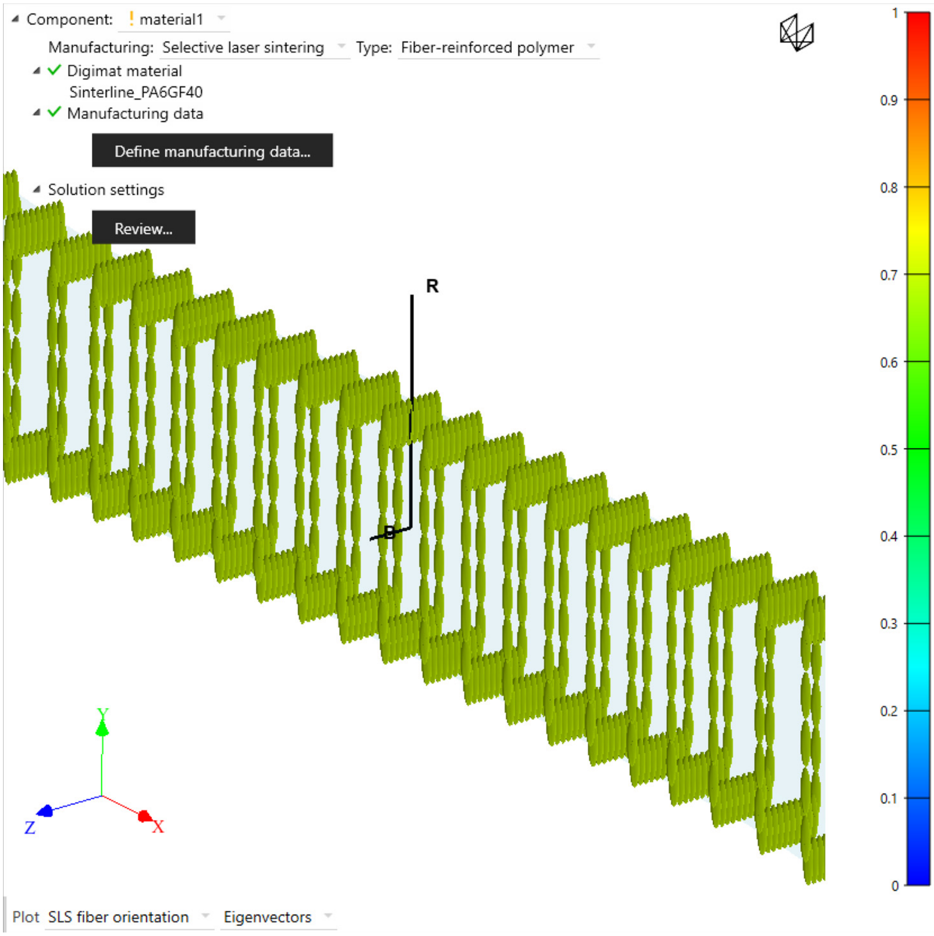

Figure 5-17 Displaying printer axis system and fiber orientation tensor from SLS manufacturing data in Digimat-RP (for “Fiber-reinforced polymer” material).

Optionally, residual stresses obtained from Digimat-AM may also be imported, requiring two files:

- Digimat-AM mesh (`*.dat`)
- Residual stresses from Digimat-AM (`*.xml`)

Once residual stresses have been loaded, a mapping step is required, as described in Fields Mapping. When the mapping step is complete, the next step in the component workflow becomes accessible: Review Solution settings.

Once residual stresses have been loaded, a mapping step is required, as described in Fields Mapping When the mapping step is complete, the next step in the component workflow becomes accessible: Review Solution settings.

CFRP Process

When using a CFRP process, the manufacturing data window is not available. Indeed, the orientation of the unidirectional or woven reinforcement is not determined using a mapped orientation file, but has to be pre-defined in the original FEA model using standard native cards. Information about how to set up these orientations can be found at Chapter 3: Coupling to Finite Element Software in *Digimat-CAE User's Guide*.

Fields Mapping

When using the “From simulation results” source for manufacturing data, a dedicated mapping is required to transfer manufacturing data from the manufacturing mesh to the component mesh used in the FEA structural analysis. Indeed both meshes are usually dissimilar (element type, element size...) as they originate from separate CAE software, and manufacturing data expressed on the manufacturing mesh is not directly usable for the component mesh. The transfer of manufacturing data to the component mesh happens via the **Map fields...** button in the component workflow. The mapping process consists of two steps: the mesh superposition and the actual data mapping. These steps are described in the following sections.

Superposition

The two meshes describing the component topology may differ in terms of position and orientation. The differences can be evaluated directly from the field mapping viewport (see Figure 5-18). To transfer manufacturing data to the component mesh, the correct superposition of both meshes is required, via adequate translations and rotations. Such geometrical operations can be automatically guessed by Digimat-RP or made fully available for a user-controlled superposition.

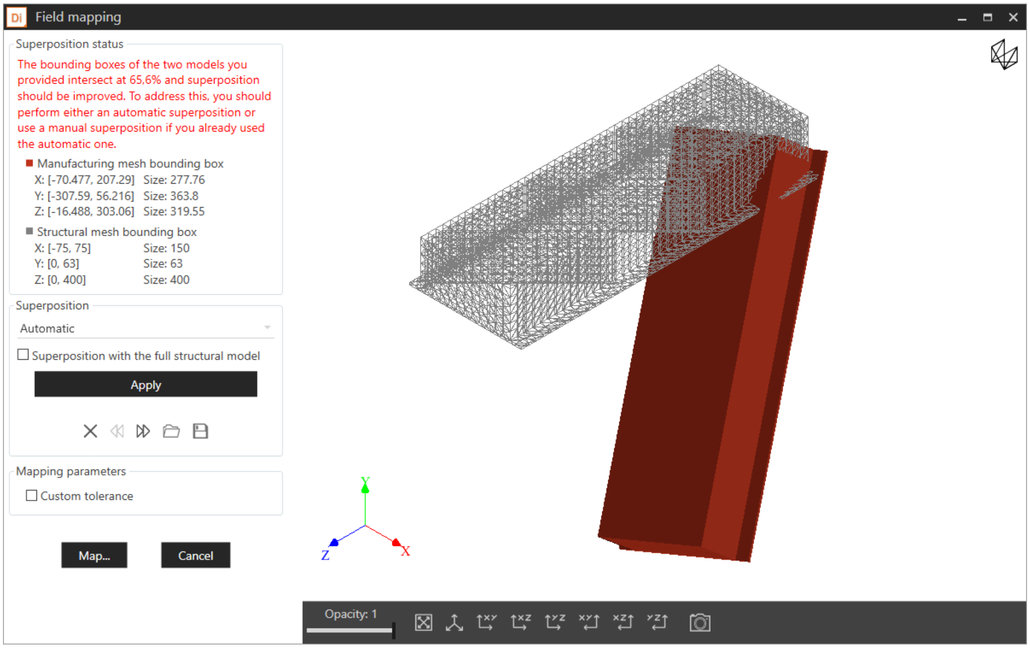

Figure 5-18 Example of initial incorrect mesh superposition.

The superposition methodology choice is provided in the Superposition group box from the Field mapping window.

When choosing the automatic superposition method, the only step required is to run the superposition via the **Apply** button. The superposition can then be visually checked again in the viewport. If further geometrical transformation is needed, then the interactive method can be used.

When using the automatic superposition method with a multi-component structural model, an additional option is available, allowing to perform the superposition between the manufacturing mesh and either the current component of the structural model, or the full structural mesh. This allows to take into account scenarios where the manufacturing mesh already contains all components, and the superposition operation should thus be done with the corresponding structural mesh.

The interactive methods allow to manually modify the position and orientation of the manufacturing mesh through translation and rotation operations, or a 'three points superposition transformation'. The scaling is performed automatically based on the defined unit systems for the structural model and the manufacturing mesh.

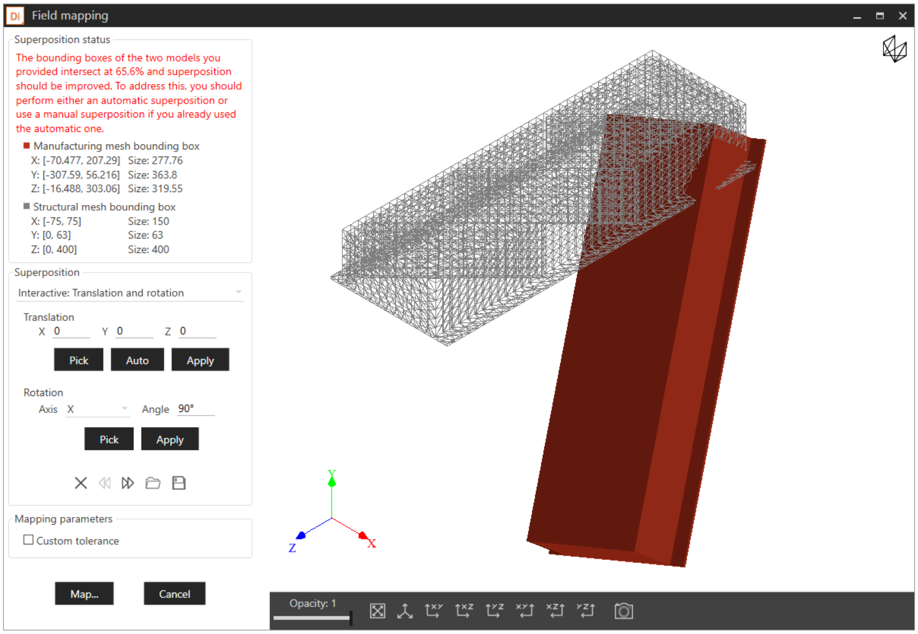

Figure 5-19 Interactive superposition controls.

When using the interactive superposition via translation and rotation operations, the transformations can be defined via their numerical values or on-the-fly by picking points from the viewport:

- Translations happen following a translation vector applied to the manufacturing mesh. The vector can either be user-defined if component values are known, or be defined by picking two equivalent nodes from the manufacturing and component meshes (in this order). The definition of these two equivalent points is accessible when clicking on the **Pick** button. Picked nodes are highlighted by spheres. Once two nodes have been picked, the corresponding translation vector is computed and can further be applied (see Figure 5-20). A third available option is to automatically suggest a recommended translation based on the current positions of the two meshes, using the button labeled Auto. This recommended translation is computed without taking into account the orientation of the meshes and aligns the centers of the bounding boxes of the two meshes. This provides a quick fix if the two meshes are identical and in the same orientation, or a useful first step otherwise, especially if the two meshes are very distant from each other.

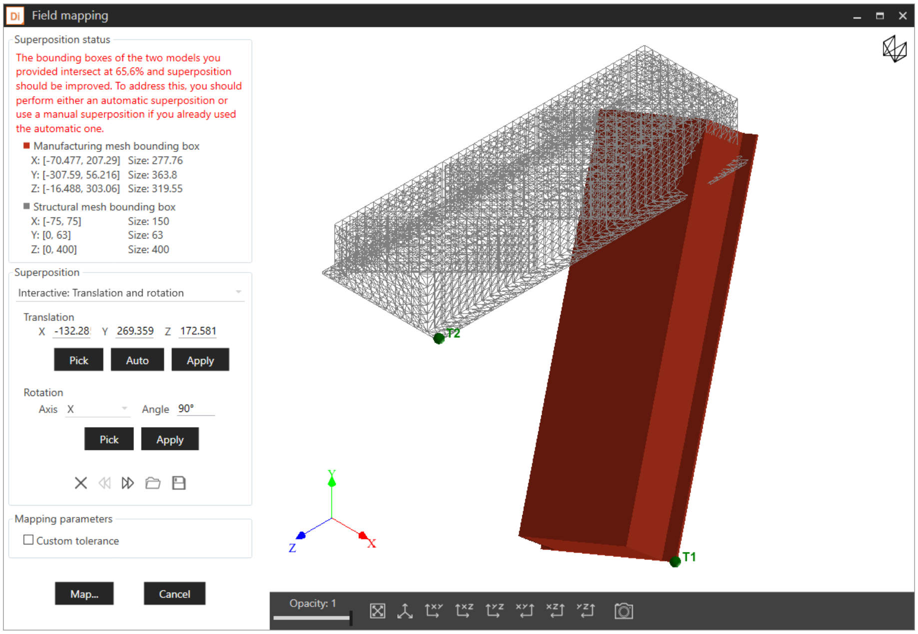

Figure 5-20 Picking two equivalent points to define translation.

- Rotations are defined by an axis and the angle of rotation around that axis. The rotation axis can be chosen as X, Y, Z or defined as a custom vector which can be obtained by selecting two nodes in the viewport. Similarly to the custom translation definition, the two nodes needed to define the custom axis can be picked from the viewport when clicking on the **Pick** button.

When using the interactive superposition via three points, a transformation combining both a translation and a rotation is performed. To define this transformation, three reference nodes are chosen on both meshes, and each node will be superposed (if possible) to its corresponding node in the other mesh. The three nodes should thus be selected in the same order in both meshes. The selection can be done using node IDs, or by picking them after clicking on the pen icon.

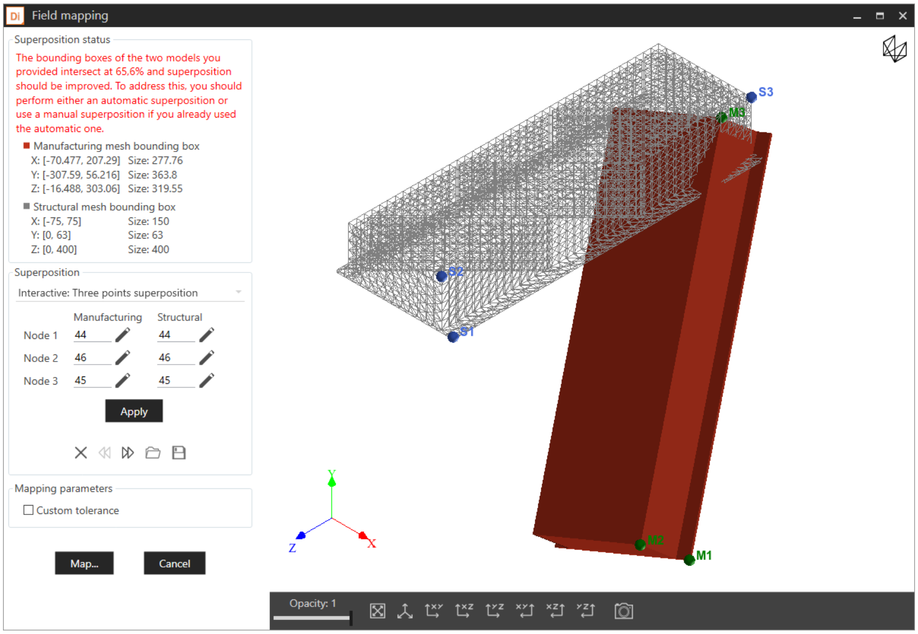

Figure 5-21 Picking three points on each mesh to define the transformation.

When the manufacturing and the component meshes are not perfectly superposed (at least 0.05% difference between the bounding boxes positions), a numerical superposition quality indicator is displayed at the top left.

The quality indicator is computed based on the volume of the intersection of the bounding boxes of both meshes. The recommendation is determined as follows:

- Intersection volume is at least 99.8%: the two models are considered as superposed.
- Intersection volume is at least 98%: the two models are close, but the superposition may be improved. It is recommended to check manually the quality of the superposition and ensure that features appearing in both models are superposed.
- Intersection volume is less than 98%: the two models are not superposed. This may happen if an incorrect unit system has been selected, or if their positions are not matching. A superposition operation should be applied.

The current positions of the bounding boxes of both meshes are also displayed below the superposition quality indicator. The three numerical ranges can help determining the exact translation to apply to superpose both meshes. When both meshes are superposed, the displayed message is updated, confirming that the current positions are satisfactory (see Figure 5-22).

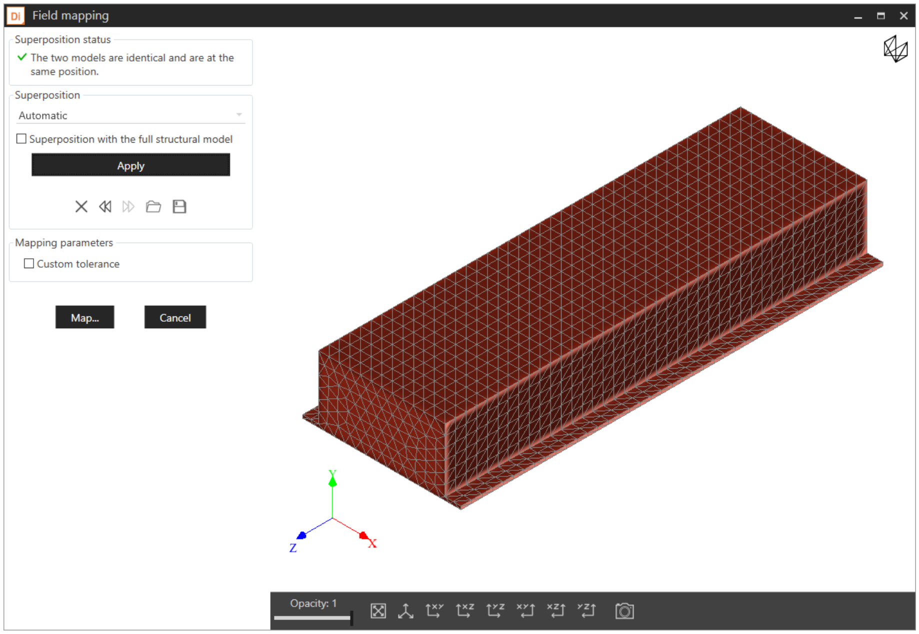

Figure 5-22   Both meshes are correctly superposed.

Whether using the automatic or the interactive superposition method, several transformation options are available from the icon menu:

- Reset: this action sets the manufacturing mesh back to its original configuration, and therefore cancels all transformation operations that have been applied
- Undo transformation: undoes the latest transformation applied to the manufacturing mesh
- Redo transformation: applies the transformation which was just undone
- Export transformation file: exports a file (`.transf` format) which saves all transformations applied to the manufacturing mesh. Please note this file can further by used in Digimat-MAP for batch mapping or scripting
- Import transformation file: allows to import a previously saved transformation in a `.transf` format and avoid to rerun all transformations manually.

Mapping

Once mapping parameters are set, the data mapping can be run via the **Map...** button. When working with several types of manufacturing data (e.g., fiber orientation and weld line), one mapping computation is launched per data type. When mapping is complete, manufacturing data can be visualized on the component mesh via the **Plot** drop-down menu in the Structural model window.

The mapping algorithm is using mapping parameters which are accessible from the user interface. While default values will usually ensure a successful mapping in most common applications, in some cases several mapping parameters may be considered prior to running mapping. These parameters and their usage are described in the next sections.

Custom Tolerance

The mapping tolerance represents the typical distance from an integration point until which manufacturing data is considered as relevant for that integration point. Increasing the tolerance may allow to capture data which is located far away from the structural model and ensure a complete mapping. However this should be performed with care as it may lead to less accurate manufacturing data for the structural model. The complete technical background of mapping tolerance is described in Mapping Tolerance.

Thickness Distribution (Shell Elements Only)

The shell layer distribution parameters can be used to reduce the number of layers in the manufacturing data file, or to change the thickness distribution pattern, for example between constant thicknesses and Moldflow mid-plane distribution. The distribution also has to be defined when performing a solid-to-shell mapping. For several FEA codes, these operations may be non optional and/or several options may be restricted, e.g., if only constant thicknesses are supported or if the maximum supported number of layers is smaller than the number of layers defined in the manufacturing data file.

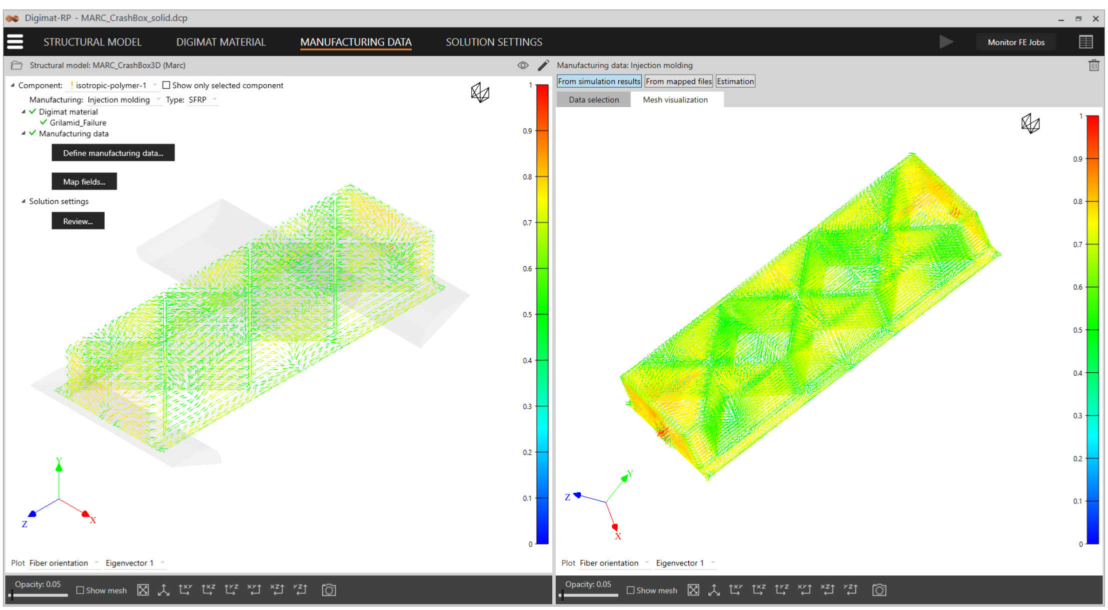

Figure 5-23 Mapped manufacturing data can be visualized on the structural model.

Uniform distribution is supported up to 20 layers or 21 integration points. Moldflow mid-plane distribution is only available when using 8, 10, 12, 14, 16, 18 or 20 layers.

When targeting an LS-DYNA shell model, it is also possible to use Gauss or Lobatto thickness distribution pattern for the integration points. These distributions are only available with at most 10 integration points. Lobatto distribution also requires a minimum of 3 integration points.

If the data to be mapped comes from 3D TIMON (3 layers with non-uniform thicknesses), the only available 1D mapping operation is to map data to 5 layers with constant thickness.

The selected thickness distribution does not need to match the distribution used in the original part/section in the structural model. Indeed, Digimat-RP will modify the assignment according to the requested thickness distribution. For the same reason, it is also not necessary to choose between thickness integration points definition with/without layers. More information about thickness distributions can be found in section Thickness Distributions.

Advanced thickness mapping parameters

When mapping data from a solid model to a shell model, the local thickness has to be computed for each element of the shell mesh. This thickness is computed based on the relative position of that shell element and the skin of the solid model. More precisely, in most cases, the intersection between the shell normal direction and the skins of the solid mesh is used as the shell element thickness. For some particular cases, such as T-joints or other non-smooth features, a more advanced process is used to determine the final thickness to use. The following parameters (see Figure 5-24) are available to customize this thickness computation for these cases, if their default values do not produce an adequate result:

- **Maximum thickness**: the maximum thickness value when computed using the shell normal and the skins of the solid mesh. In Digimat-RP, the default value is set to the maximum thickness detected in this component in the original model.
- **Maximum angle between shell normal and solid skin normal**
- **Maximum thickness ratio between connected elements**

Details about these three parameters can be found in the section Solid to Shell Mapping of Digimat-MAP User's Guide.

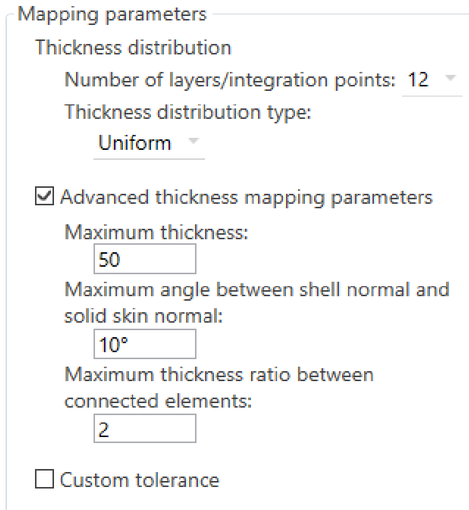

Figure 5-24 Advanced parameters used to customize the thickness computation when mapping data from a solid model to a shell model.

Additional settings

Depending on the type of data to map, additional settings may be available (see Figure 5-25 and Figure 5-26).

- When mapping fiber length data without fiber diameter data nor aspect ratio data, the fiber diameter to use can be defined. By default, the value used is the average fiber diameter, computed from the distribution of fiber length loaded on the manufacturing mesh, and the fiber aspect ratio defined in the Digimat material.
- When mapping weld line data, a weld line radius can be set to extend the area of the component mesh to be considered as affected by the weld line, after mapping data from the manufacturing mesh.
- When mapping weld line angle/criterion data, additional bounds can be defined so that only points with a weld line angle/criterion inside the bounds are taken into account for the mapping.
- In the Fused filament fabrication and Fused deposition modeling processes, settings related to toolpath mapping are available. If the toolpath file contains bead width information, it will be automatically taken into account. Otherwise, the bead width must be specified manually. Lattice infill mapping is also available.

More information can be found in section Weld Line Mapping for the weldline mapping and in section Lattice Infill for the lattice identification.

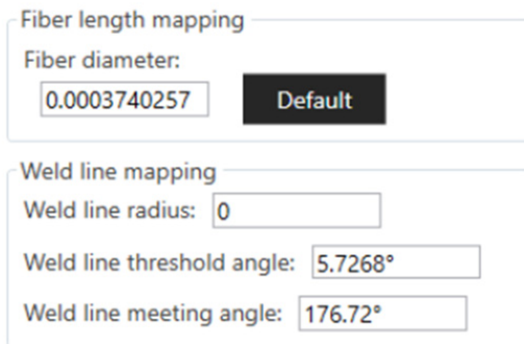

Figure 5-25 Additional settings when mapping fiber length data or weld line angle data.

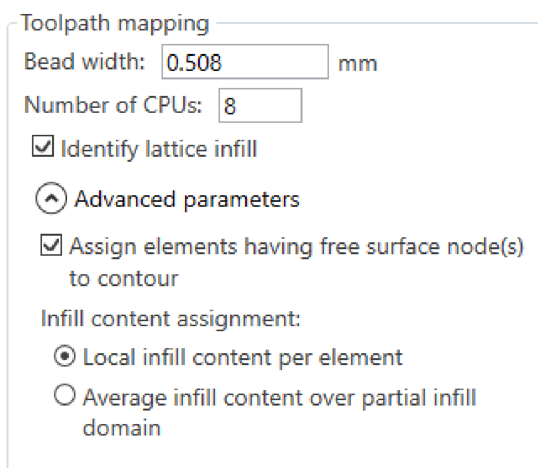

Figure 5-26 Additional settings for lattice identification in FFF and FDM manufacturing process.

6 Solution Settings Window

- Solution Settings
- A Special Case: The Macro Solution

Solution Settings

The Solution settings window is available at the end of a component workflow and is divided into two tabs, **Solution settings** and **Advanced solver settings**. It enables to review the numerical settings for the analysis as well as to choose the coupling solution method to be used for the FEA run. The window can be accessed from the Solution settings tab in the top ribbon, or directly from the Digimat material window by switching to the Solution tab.

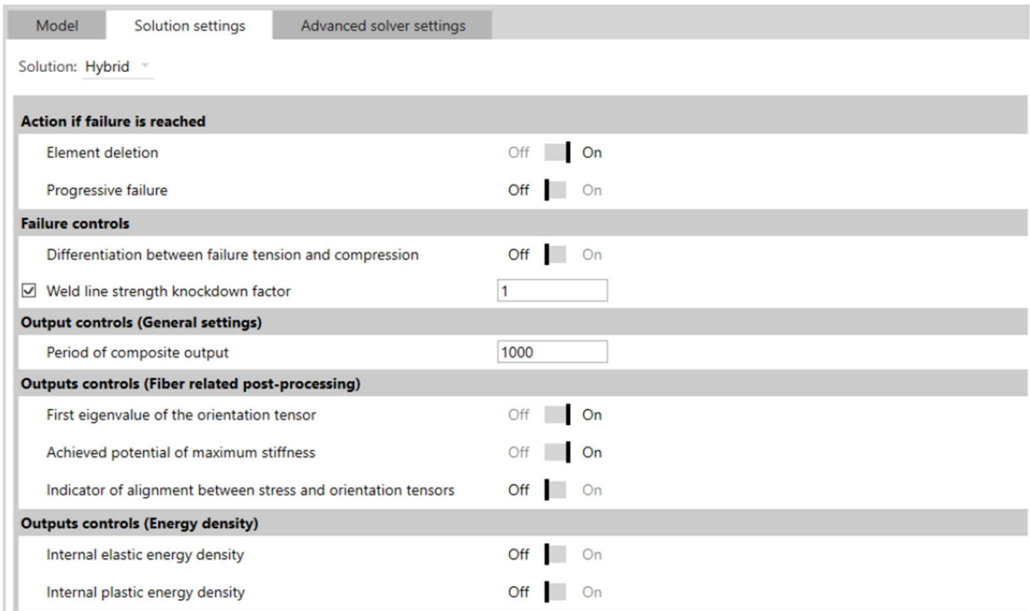

Figure 6-1 The Solution settings tab.

The two tabs gather all parameters that are not related to the composite material physical definition. The "**Solution settings**" tab contains all high-level parameters related to the analysis while the "**Advanced solver settings**" tab contains advanced parameters to fine-tune the behavior of the material model. The "Solution settings" tab also allows to choose the solution procedure to use (e.g., Hybrid, Micro, Macro). For more details about the different solution procedures, see section Solution Methods.

Parameters with a checkbox in front of them have the following behavior:

- If the checkbox is left unchecked, the default value will be used.
- If the checkbox is checked, a user-defined value will be used instead.

For more details about the different parameters, see section Analysis Parameters.

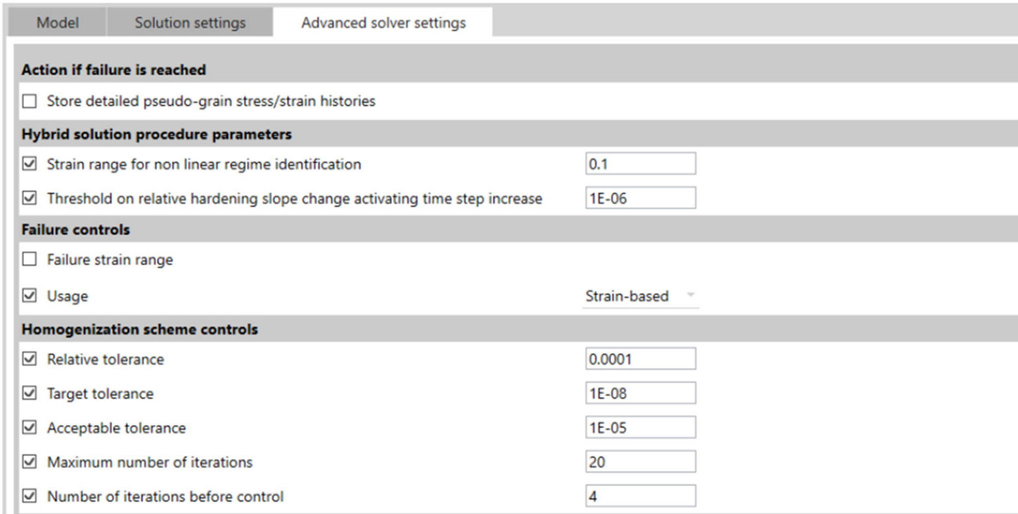

Figure 6-2 The Advanced solver settings tab.

Guidance for Typical Applications

A few parameters are worth noting when working on some typical applications. The next sections highlight several typical applications of Digimat-RP and the related solution settings of interest. In a general way, it is highly recommended to select the Hybrid solution procedure. The next sections assume that it is systematically the case.

Failure Modeling

When working on failure modeling in general, several parameters are key to the application definition:

- Stop analysis (when failure is reached): this setting is only available for implicit FEA analysis using the Hybrid solution procedure, and cannot be combined with element deletion (see below). Activating this setting enables to stop the analysis when the failure indicator reaches a critical value (1 by default) at any material point. Note that the default critical value is often very conservative, so it may be user-defined to a higher value.
- Element deletion: activating this parameter enables to study the consequences of failure initiation on the structural model response. Once final failure (which is considered as reaching failure initiation when no progressive failure is applied, or reaching maximum damage when using stiffness reduction) is reached in an element of the FEA analysis, the element will be deleted by the FEA solver. While using element deletion in explicit crash simulation is standard practice, please note that for implicit simulation this is an advanced failure modeling option of Digimat requiring care in the FEA model setup, as explained in the Digimat-CAE guidelines (see subsection Analysis Parameters & Convergence Issues).

- Progressive failure: activating this parameter enables to study the consequences of failure initiation on the structural model response. Once failure is reached in an element of the FE analysis, stiffness will be progressively reduced to account for the damage in the material. Please note that this is an advanced failure modeling option of Digimat requiring care in the FEA model setup, as explained in the Digimat-CAE guidelines (see subsection Analysis Parameters & Convergence Issues).
- Damage law: this option appears when progressive failure is selected. The user as the choice between Instantaneous, Linear and Power law (see section Failure parameters for more information)
- Damage equivalent strain between failure initiation and final failure: this option appears when the Linear or Power law damage law are selected. It drives the amount of equivalent strain during which damage occurs.
- Power law exponent: this option appears when the power law damage law is selected. It controls the shape of the damage slope.
- Maximum damage: this option appears when progressive failure is selected. It controls the maximum damage that the material can accumulate. When this value is reached, the stiffness remains constant. It also trigger element deletion when this option is activated.
- Time step multiplier for failure propagation: this option appears for Abaqus/Std and Marc when progressive failure is selected. It allows the user to impose a smaller time step from Digimat when failure is initiated.
- Differentiation between failure strain in tension and compression: toggling this option allows to easily account for a difference between tensile and compressive failure when the Digimat material model loaded in Digimat-RP only contains a tensile calibrated failure model. This option only appears on non-encrypted material model.
- Ratio between compressive and tensile failure strains: defining this parameter allows to easily control the difference between tensile and compressive failure. A value of 1 means no differentiation, while a value of 3 means that strain at failure in compression is three times higher than the strain in tension. This option only appears on non-encrypted material model.
- Failure computation methodology: this option only appears for Fused Filament Fabrication and Fused Deposition Modeling. It drives the failure of a material point from the failure of the individual filament orientation.

Weld Line Strength Modeling

When working on a molded application (injection or compression molding), the impact of weld lines on the component strength may be evaluated. Weld line areas can be a source of component failure, not only because of the local specific fiber orientation due to the several melt fronts meeting in the weld line area, but also because material suffers from local degradation due to defects, dusts, excessive local heating... This local strength degradation in the weld line area can be specified in the **Solution settings** tab.

Once weld line manufacturing data has been defined for the component (i.e., after mapping weld line data, or loading a mapped weld line file), the weld line strength can be specified in the Solution settings via the "Weld line strength knock-down factor" parameter. This parameter locally applies a multiplicative knock-down factor on the composite material strength during the FE analysis as soon as an element is localized in the mapped weld line region. As an example, a value of 1 means no degradation while a value of 0.75 means the material strength in the weld line area is reduced by 25 percent. As a result, early failure in the weld line area can be captured thanks to the contribution of both the fiber orientation and weld line degradation effects.

A Special Case: The Macro Solution

In this section, information related the macro solution is available. Its capabilities, solutions and the issues that it can cause are also discussed.

Capabilities

On top of the three solutions available in Digimat-CAE to interface structural FEA software with Digimat (see section Solution Methods), Digimat-RP offers a fourth method called the Macro solution. It makes it possible to couple Digimat with some structural FEA software without any preexisting integration of Digimat-CAE:

- Abaqus
- ANSYS (for solid elements only)
- LS-DYNA
- MSC Nastran (Macro solution sequences SOL1XX only)
- OptiStruct
- PERMAS

Digimat-RP works with these types of models just like with any other type of structural model. The main difference is that the Macro solution is the only coupling method available for MSC Nastran SOL1XX, OptiStruct and PERMAS models.

Currently, the Macro solution only supports:

- For Abaqus
  - shell or solid elements;
  - (thermo-)elastic material models;
  - (thermo-)elasto-plastic material models;
  - (thermo-)mechanical analyses;
  - two-phase materials
  - Standard analysis (explicit is not officially supported);
- For ANSYS:
  - solid elements;
  - (thermo-)elastic material models;
  - (thermo-)elasto-plastic material models;

    - (thermo-)mechanical analyses but excluding damping effects;
    - two-phase materials
- For LS-DYNA:
    - shell or solid elements;
    - elastic material models;
    - elasto-plastic material models;
    - elasto-viscoplastic material models;
    - simple failure model based on maximum principal strain in tension and in compression for LS-DYNA R9.3 and R10.2 (solid elements);
    - explicit mechanical analysis;
    - strain-rate elasto-plasticity;
    - Tsai-Wu failure model for LS-DYNA R10.2 (shell elements) and LS DYNA R11.1.
- For other CAE codes:
    - shell or solid elements;
    - linear mechanical analyses (excluding thermal expansion phenomena) such as linear static, modal and frequency response analyses;
    - linear thermo-mechanical analyses but excluding damping effects;
    - either two-phase, elastic Digimat materials (OptiStruct, PERMAS and Abaqus models) or two-phase, elastic or viscoelastic Digimat materials (Nastran SOL1XX models);– For PERMAS, elastic card with fatigue criteria are supported but the fatigue criteria are not used in the PERMAS simulation.

A more detailed list of limitations can be found in section Using the Macro Non-Linear Solution.

Remark: The Macro solution is not available when using the legacy manual workflow to generate the interface file.

Description of the Solution

The Macro solution only achieves a weak coupling between Digimat and the finite element model. More particularly, the macroscopic properties of the Digimat materials are pre-computed prior to the finite element analysis which then runs without Digimat (re-)computing anything as the solution proceeds. To the contrary of the Macro solution,

- the material properties are explicitly written to the input files of the coupled analysis instead of being communicated live by Digimat;
- the coupled model can run without Digimat, with a classical installation of the structural FEA software.

The modifications Digimat-RP makes to the structural model only affect the material properties and the coordinate systems as described below. The other data in the structural model remain unchanged.

The degree of refinement of Macro solution can be directly controlled from the Linear template. Five levels of reduction are possible:

- highest: at most 6 material properties
- high: at most 15 material properties
- medium: at most 45 material properties
- low: at most 105 material properties
- lowest: at most 231 material properties

By choosing a high level of reduction a few material properties will be written in the FEA input deck, corresponding to a few typical fiber orientation states. By using a low level of reduction, a high number of material properties will be written in the FEA input deck, describing in the most accurate way the variation of fiber orientation throughout the component.

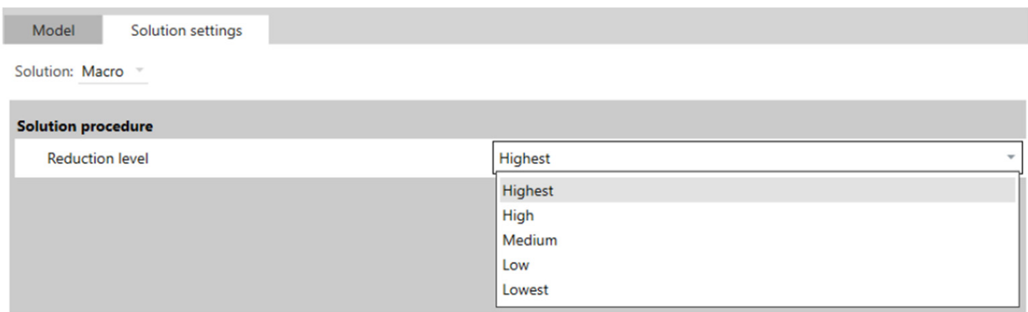

Figure 6-3 Reduction level selection from the Macro template.

Material orientations

In case the user requests the fiber orientation to be taken directly from the Digimat material (see section Component Selection), Digimat-RP will assume this fiber orientation is defined with respect to the material coordinate system of the elements, as defined in the input structural model. If there is no material coordinate system defined for these elements in the structural model, not even a default one, Digimat-RP will define one which is aligned with:

- the global Cartesian coordinate system for solid elements;
- the direction going from the first node to the second node in the definition of the element for shell elements.

It is therefore the user's responsibility to make sure the material orientations are adequately defined in the input model to get the desired behavior.

On the contrary, if the user requests the fiber orientation data to be obtained from the manufacturing process or from a pre-existing orientation file (see section Component Selection), Digimat-RP will take these information into account by defining:

- a new, specific coordinate system for each solid element to which a Digimat material is assigned;

- a new, specific layered composite material with particular ply orientation angles for each shell element to which a Digimat material is assigned.

Stiffness Properties

Digimat-RP will typically add between 20+ to 200+ new material properties to the input model. All together, they are meant to represent with sufficient accuracy the homogenized stiffness behavior of the composite material for all possible distributions of fiber orientation. Number of added material properties is inversely proportional to the reduction level requested for the solution in Digimat-RP. These material properties will then be assigned to the different solid elements or to the different layered composite materials mentioned in the preceding section. In case the orientation is taken from the Digimat material and is therefore spatially uniform, only one out of the 200+ material properties is needed and gets written to structural model modified by Digimat.

Damping Properties

When using elastic Digimat materials (all CAE codes, except LS-DYNA and ANSYS), the structural damping coefficients appearing in the input model are assigned to the newly created material properties. Of course, the resulting damping effect will be different since the Digimat material properties most likely have different stiffness properties than the materials initially present in the structural model.

When using viscoelastic Digimat materials (which is only supported with MSC Nastran SOL1XX models), the viscous damping of the materials will replace any structural damping coefficient initially present in the structural model. With multi-phase materials, the corresponding damping matrix generally does not scale with the stiffness matrix. In MSC Nastran version 2017.0, there is no built-in feature making it possible to define such a damping. To define the right anisotropic damping at a particular frequency, Digimat-RP therefore:

- assigns zero damping properties to the elements initially present in the structural model (they will contribute only to the global mass and stiffness matrices of the finite element problem);
- duplicates these elements and assigns to each of these duplicates specific properties adding significant contributions to the global damping matrix of the finite element problem but negligible contributions to the global mass and stiffness matrices.

Just like for the stiffness properties, these damping properties rely on 20+ to 200+ newly created materials which span with sufficient accuracy the full range of potential distributions of fiber orientation and which show significant, anisotropic damping properties but negligible stiffness or mass properties. These duplicate elements and the associated materials are only added to the MSC Nastran SOL1XX solution sequences supporting damping, i.e. SOL 107-112, 118, 129, 145-146, 159 and 200.

Starting from MSC Nastran version 2018.0, the MAT2 and MAT8 cards were supplemented with extra anisotropic structural damping entries making it possible to define separately the different components of the structural damping matrix rather than obtaining it by scaling the stiffness matrix. To define the right anisotropic damping at a particular frequency with MSC Nastran version 2018.0 or newer, Digimat-RP hence simply includes in the definition of the 20+ to 200+ new materials the appropriate anisotropic structural damping coefficients. These are only added to the MSC Nastran SOL1XX solution sequences supporting damping, i.e. SOL 107-112, 118, 129, 145-146, 159 and 200.

Frequency Dependent Properties

Viscoelastic materials, which are only supported in MSC Nastran, exhibit rate-dependent and, hence, frequency dependent stiffness and damping properties. This dependence can be taken into account by Digimat-RP in the case of frequency response analyses (SOL108 and SOL111). The way it is handled depends upon the MSC Nastran version which is targeted and which therefore needs to be specified by the user (see section Submitting Jobs).

For MSC Nastran 2018.0 or newer The frequency dependence of the properties of the material is taken into account with the help of specific cards (`MAT1F`, `MAT2F`, `MAT8F` and `MAT9F`) newly introduced in MSC Nastran 2018.0. Only one input deck and one MSC Nastran run are then needed to compute the frequency response. For SOL111 i.e. modal frequency response analyses, the eigenmodes used to decompose the solution are computed with the nominal, frequency-independent properties defined in the `MAT1`, `MAT2`, `MAT8` and `MAT9` cards and supplemented with extra residual vectors accounting for the frequency-dependent behavior defined in the `MAT1F`, `MAT2F`, `MAT8F` and `MAT9F` cards. This solution is more computationally efficient at handling the frequency dependence than the solution. It typically reduces the computational time by 15 to 75%. Besides, with this solution, SOL111 analyses can typically run 10 times faster than the corresponding SOL108 analyses.

Computational efficiency aspects Accounting for the frequency dependence of the material properties in a frequency response analysis definitely gives more accurate results but typically requires longer computational times. A trade-off needs to be made between computational efficiency and accuracy. When using MSC Nastran 2018.0, the simulations typically are 1.1 to 5 times longer but these numbers can easily be multiplied by 10 when working with earlier versions of MSC Nastran. The user is therefore offered the possibility to neglect the frequency dependence of the material properties. In that case, constant material properties corresponding to a particular frequency to be specified by the user are used in the analysis. See section Submitting Jobs for more details.

Note:	The above methodology has some important implications which are listed below.

For all types of input models

- After being coupled with one or more Digimat materials, the structural model will include several hundreds of material properties. It is therefore not recommended to reload this model in Digimat- RP to edit or modify the way it is coupled with Digimat materials. The user should rather reload the Digimat complete project which was created to generate the first structural model coupled with Digimat. It requires saving that project before closing Digimat-RP, see section File Group.
- The total shell thickness initially defined in the structural model gets only modified when using an orientation file defining the absolute thickness of the different layers. In all other cases, that total thickness is preserved. Only its distribution over the different layers can potentially be modified when using an orientation file defining the relative thickness of the different layers.
- New materials added by Digimat-RP are based on redefined orientation. Therefore, when looking at the stress results, do not forget to first transpose them into a global coordinate system for a correct interpretation.
- For CAE codes supporting nonlinear material behavior:

- Material cards will also be encrypted when original Digimat material file is encrypted and/or if material behavior is nonlinear (EP/EVP/TEP).
- Generation of nonlinear material cards needs a special license feature `EX_DIGIMAT_CAE_MACRO_NONLINEAR`.

For all supported CAE codes that have strong coupling interface with Digimat

To run model coupled with multiple material models, some of them using hybrid/micro solution and others using Macro solution, it is needed to work in 2 steps:

1. Generate models with hybrid/micro solution in a given working directory
2. Select Digimat-RP **Copy files...** instead of **Run locally**, then **Dismiss** after copying the files (see Figure 6-4)
3. Start a new Digimat-RP project and import the Digimat coupled input deck located in first directory.
4. Select Macro solution and set working directory to directory where files have been copied.
5. Generate Macro material properties and run your computation.

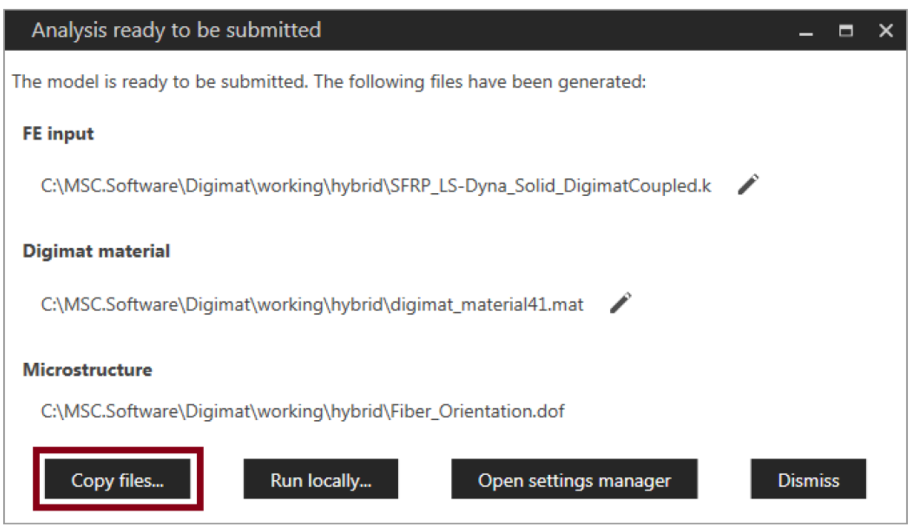

Figure 6-4 Copy ready-to-run files using hybrid/micro solution.

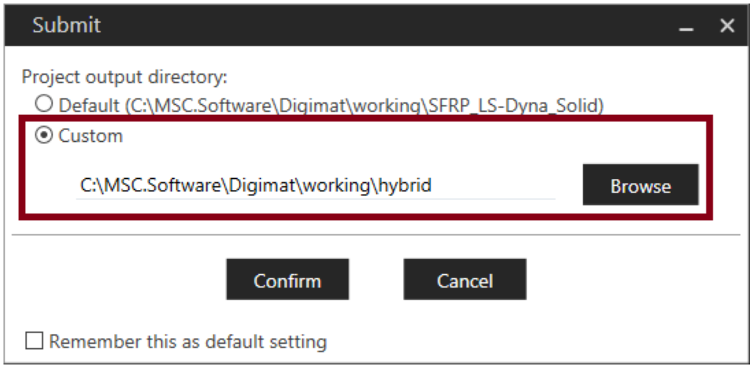

Figure 6-5 Custom working directory.

For both MSC Nastran SOL1XX and OptiStruct models

- Any global damping definition appearing in the structural model – whether structural (parameter `G` or `GFL`), modal (Case `SDAMPING`) or Rayleigh (parameters `ALPHA1` and `ALPHA2`) – is preserved. The resulting damping effect will be affected by the Digimat materials since these will most likely modify the stiffness properties and the eigenmodes of the structure.
- Any `PCOMP` initially making use of a material to be replaced by a Digimat material without any orientation data will, after coupling with Digimat, refer to a `MAT2` in its definition and therefore have zero transverse shear flexibility.
- The `PCOMP` properties which are created in replacement of a `PSHELL` property will only include membrane or bending terms if the initial `PSHELL` property only includes membrane or bending terms.
- The `PCOMP` properties which are created in replacement of a `PSHELL` property will not yield any stress or strain output for the individual plies (field `SOUTi=NO` in the `PCOMP` card). To override that behavior, the user needs to switch to `PCOMP` properties in the initial structural model and explicitly set `SOUTi` to `YES` wherever needed.

For MSC Nastran SOL1XX models only

- A model with Digimat materials initially targeting MSC Nastran 2017.0 or 2017.1 will run perfectly fine with any newer version of MSC Nastran.
- A model with Digimat materials initially targeting MSC Nastran 2018.0 or newer will not necessarily run with older versions of MSC Nastran.

For PERMAS models only

- The `$LAMINATE` materials created by Digimat-RP and assigned to shell elements are defined with plies which can all carry membrane, bending and transverse shear loads. The only way to have these shell elements carry either only bending and transverse shear loads or only membrane loads is to assign the proper `$GEODAT CNTRL` properties to the elements in the input structural model. It is in no way possible to define ply-by-ply which loads can be carried for the `$LAMINATE` materials created by Digimat-RP.

- Any TRIA3, TRIA3K, TRIMS3, TRIM3 element or any SHEAR4, QUAD4, QUAMS4, QUAM4 element to which a Digimat material with orientation data is assigned will be converted to a SHELL3 or SHELL4 element respectively because the latter types of elements are the only ones which layered composites ($LAMINATE) can be assigned to.
- Any geometrical data ($GEODAT) assigned to the elements in the input structural model will remain in the model after the modifications made by Digimat-RP.
- The $LAMINATE materials in PERMAS do not support material dependent properties (ex. Temperature dependent properties). Therefore, if the replacing material has temperature dependent properties and is applied on an existing $LAMINATE or will be associated to created $LAMINATE if orientation file is associated, the Digimat coupled model will not be generated. Note that thermo-mechanical properties independent of the temperature are allowed.

For Abaqus models only

- Digimat-RP will replace *SOLID SECTION or *SHELL SECTION of the original model when associated material is replaced by a Digimat material. Elements in section's ELSET are reorganized in new *ELSET corresponding to the Digimat material to which they will be associated and a new section is created, linking the new material and element set. Other section as well as all pre-existing *ELSET are kept in the model. Original section options are applied to new sections, except SYMMETRIC (all layers are written in new shell sections) and LAYUP in shell sections (unsupported).
- When *HOURGLASS STIFFNESS command is specified in the replaced section (due to usage of reduced elements), the command is also added into the new section created by Digimat-RP with recomputed hourglass stiffness values.
- When *TRANSVERSE SHEAR STIFFNESS command is specified in the replaced section, the command is also added into the new section created by Digimat-RP with recomputed transverse shear stiffness values.
- If replaced original material has a *DAMPING command, it will be applied on all new material cards generated by Digimat-RP, using the same values.

- Linear solution for abaqus in Digimat-RP does not support input model where shell or solid properties are defined by means of auxiliary data such as *DISTRIBUTION and *DISTRIBUTION TABLE.

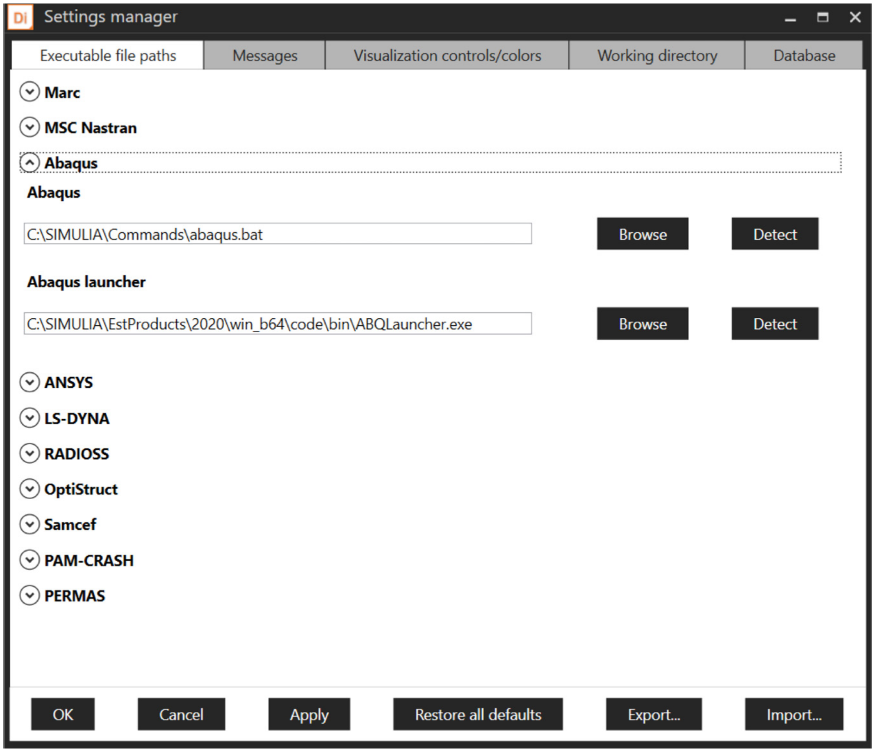

Figure 6-6 Setting Abaqus-CAE path for encryption of material cards.

- If initial (T)EP material model is encrypted, material card will also be encrypted. To that goal, it is needed to have a local Abaqus installation. This local Abaqus installation must be defined in Digimat- RP settings in **Abaqus Launcher** section (see Figure 6-6). This setting is automatically created at Digimat installation when defining Abaqus-CAE installation directory.

For LS-DYNA models only

- Original *`PART` command will be replaced by *`PART_COMPOSITE` command. Following configurations are not supported:
    - Lobatto and Gauss integration rules
    - Nodal thicknesses
    - `*INITIAL_STRESS` cards
- When using encrypted Digimat material with elasto-plastic behavior, all LS-DYNA material properties are encrypted during material properties generation. They are not directly readable by the user. In that goal, an encryption key file must be generated when installing Digimat. This key file must be located in Digimat working directory defined during Digimat installation. If this file

is renamed or moved to another location, the corresponding key `LSTC_PGPKEY` of `DIGIMAT_Settings.ini` file must be changed accordingly. Procedure to generate encryption key file is described in section Generation of Encryption Key File for Macro Solution *in the Digimat Installation and Operations Guide*.

- It can be needed to increase memory when using Macro solution (memory parameter on *`KEYWORD` card).
- For LS-DYNA R9.3 and R10.2 (solid elements), Principal strain at failure is suggested based on Random 3D (for solid elements) or Random 2D (for shell elements) orientation based on curves plotted in Material component. If a traction/compression differentiation factor is defined in Digimat analysis file, it is applied to the suggested value (see Figure 6-7).

Figure 6-7 Setting failure parameters for LS-DYNA macro computations.

- For LS-DYNA R10.2 (shell elements) and LS-DYNA R11.1, failure criterion are computed in a element-by-element way. In this case, for material depending on strain rate, user has to specify the average strain rate of the simulation to get to most accurate value of the failure parameters, as illustrated in Figure 6-8.

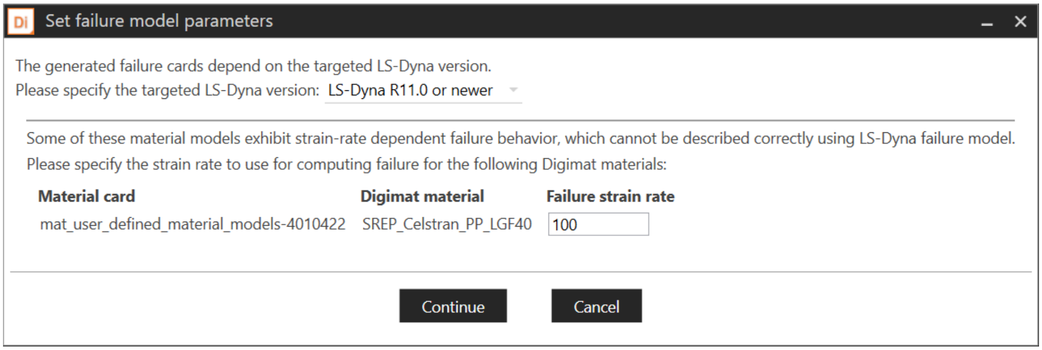

Figure 6-8 Definition of average strain rate for strain rate dependent materials.

For ANSYS models only

- Only solid elements and parts are supported.
- All the new properties (e.g. ANSYS APDL commands) related to the Digimat material are written in a separate input deck, which is then included in the main ANSYS input deck (just before the very first material command). This means that the main input deck is barely modified.
- When using encrypted Digimat material with (thermo-)elasto-plastic behavior, all ANSYS material properties are encrypted during material properties generation. These encrypted properties are then written to a second include file, included just after the first one. The encryption process can only be done by starting an ANSYS service, and consequently:
    - It is mandatory to define a valid path to the ANSYS installation folder, using the `ANSYS_Directory` of `DIGIMAT_Settings.ini` file.
    - The encryption process requires to briefly consume an ANSYS license during the material properties generation (see Figure 6-9).
- Please have look at the related potential issues and their workaround in Potential issues when generating the inputs.
- When using Digimat material with thermo-elastic or thermo-plastic behavior, the input file must contain the TREF command, which defines the environment temperature (e.g. the temperature at which thermal strains are equal to zero).
- Include files (e.g. \INPUT commands) are not directly supported in the starting ANSYS input deck. However, it is possible to run the Digimat-RP workflow on a portion of a bigger model (e.g. an include file). In such case, Digimat cannot automatically determine the starting values for the IDs of the new material properties and coordinate system that need to be introduced in the model. This means that the user must manually specify them (see Figure 6-10), and make sure that they will not conflict with existing IDs in the whole model.

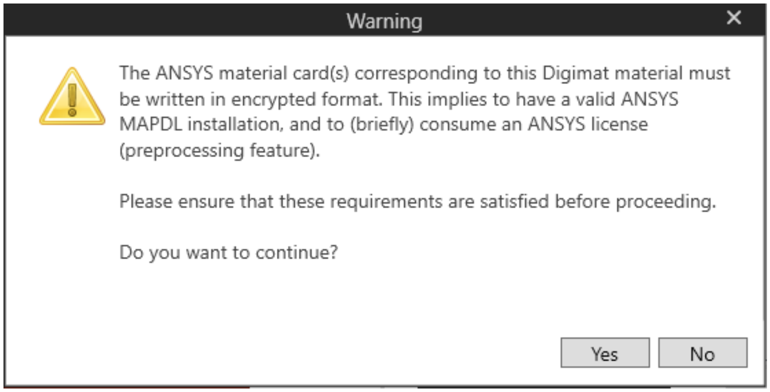

Figure 6-9 A valid ANSYS license is required to proceed with encrypted (T)EP materials.

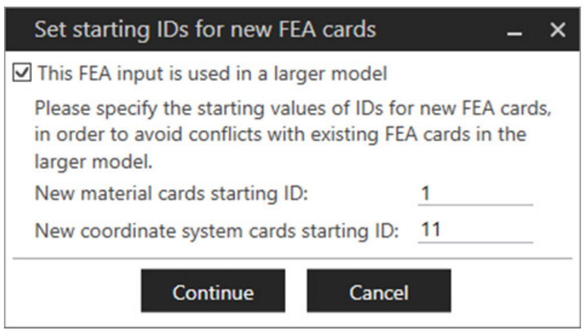

Figure 6-10 Specifying starting values for the material and coordinate system IDs.

Potential Issues

The macro solution might present some issues with some products. These potential issues are discussed in this section.

Potential issues when generating the inputs

With ANSYS Models

A common source of issues when using (thermo-)elasto-plastic materials is the ANSYS encryption process. When dealing with a distant ANSYS license server or a distant ANSYS installation, the encryption processed may fail due to a slow or non-responsive licensing system, or simply because all the licenses are already consumed. Shall this happen, ANSYS will display a message, but will unfortunately not terminate itself; Digimat will then kill the ANSYS process (and all its children) after a given number of seconds (60 by default), and an error message will be displayed in Digimat-RP. Note that this default Digimat-side timeout value can be modified through the `ANSYS_encryption_timeout` key in the `DIGIMAT_Settings.ini` file. It should be defined in agreement with the ANSYS license timeout settings (if defined).

With LS-DYNA models

- GnuPG is not found when trying to encrypt material properties. Check that the file

  `C:\Program Files (x86)\gnupg\bin\gpg.exe`

  exists and that the setting `GnuPG_Directory` points to the directory containing `gpg.exe` file. If the file does not exist, it is needed to re-install GnuPG.
- Encryption key is not found. Check that the file mentioned by the `LSTC_PGPKEY` entry of `DIGIMAT_Settings.ini` exists. If not, it needs to be generated once after the installation of Digimat and put in the directory mentioned by the `LSTC_PGPKEY` entry (see Generation of Encryption Key File for Macro Solution *in the Digimat Installation and Operations Guide*).

With Abaqus models

When using encrypted materials, path to `ABQLauncher.exe` file must be correctly set in the Digimat-RP settings manager.

Potential issues when running the job

With Abaqus Models

If the orientation of the Digimat material is taken from the Digimat material itself (i.e. no orientation data), the corresponding orientation tensor must be diagonal.

With PERMAS Models

When running a model using the defaults of PERMAS installation, it the job may fail with the following error:

```
*W* 101/1005 Requested Memory XXXX (MiB) exceeds Virtual Memory YYYY (MiB).
```

This is because PERMAS tries to allocate a memory buffer of pre-defined size, which may be higher than the currently available memory. The PERMAS Control Center provides a direct way to specify this predefined buffer size, but Digimat-RP does not. Should this issue occur the workaround is to edit the `PERMAS_DYN_BLK` variable in the following PERMAS configuration file (may require Administrator mode):

```
INTES_INSTALL_DIR\V17\PERMAS \S64\etc\permasrc
where INTES_INSTALL_DIR is the INTES installation directory.
```

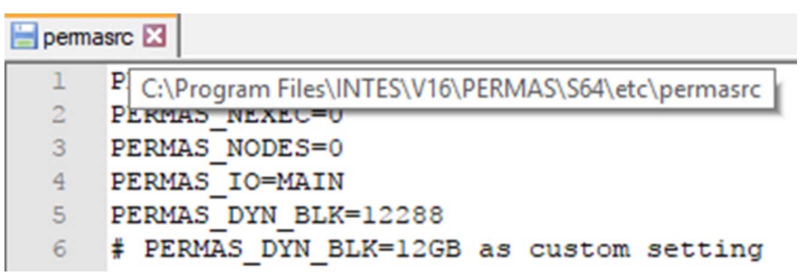

Figure 6-11 Setting the default memory size for PERMAS solver.

When running a model coupled with Digimat, PERMAS can sometimes issue warning messages of type 605/2015:

```
*W* 605/2015 Element <see list> (SHELL3/SHELL4): Element thickness defined via
laminate
description, thickness from geometrical data input ignored.
```

This happens when a Digimat material with orientation data is assigned to TRI*3 or QUA*4 elements but can safely be ignored.

With MSC Nastran SOL1XX models With shell elements, the user might experience under some circumstances an increase of the computational time by a factor of more than two. This will force MSC Nastran to use a different preprocessor which is less permissive in terms of syntax than the default preprocessor. Therefore, the user first needs to make sure the initial structural model runs properly with that extra command.

Under some circumstances, when running a model coupled with Digimat, MSC Nastran can issue warning messages of type 6064. If these messages are not issued when running MSC Nastran with the initial structural model, they can safely be ignored.

Potential issues when post-processing the results

With ANSYS models When comparing in ANSYS Workbench or ANSYS Mechanical APDL the results obtained by the Macro (Non-)Linear and Hybrid, the following issues can be observed:

- When comparing "equivalent" Von Mises measures of the stress or strain tensors, significant differences can be observed between the two solution methods, whereas all the tensor components (and the principal components) do match.
- When using thermo-mechanical models, the "Elastic strain" and the "Thermal strain" measures will not match with both solutions either. This is because the Hybrid solution does not make the distinction between the mechanical and thermal strain, while Macro (Non-)Linear does. Consequently, it is advised to use the total strain ("ETOT"), available in ANSYS Mechanical APDL only, to compare the Macro (Non-)Linear and Hybrid strain results.
- A consequence of the two previous points is that neither "Elastic strain components" nor the "Equivalent total strain" are valid measures for comparing the two solution methods.

7 FEA Jobs Submission and Monitoring

- Submitting Jobs
- Running and Monitoring Jobs

Submitting Jobs

The end of the workflow is reached when the following conditions are all satisfied:

- a structural model or a structural results has been loaded.
- at least one Digimat material has been defined and associated to a region of the structural model.
- all defined Digimat materials have been completely validated.
- all regions associated to all defined Digimat materials have required manufacturing data, whether it comes from a manufacturing process, a mapped file or from the material file itself.

The **Submit** button in the **Home** ribbon is then activated and allows you to start the generation of the coupled analysis files. This process produces the following files:

- the input deck associated to the structural model, modified with the specified Digimat materials and manufacturing data,
- one `.mat` file for each Digimat material used in the analysis (except when targeting Nastran, OptiStruct, PERMAS, Abaqus or LS-DYNA using the Macro solution),
- a copy of relevant manufacturing data files.

When generating an interface file through the legacy manual workflow, the submission requirements are simplified to:

- the structural model format has been defined.
- a Digimat material has been defined and validated.
- required manufacturing data has been defined (from a mapped file or from the material file itself).

The **Submit** button is then activated and allows you to start the generation of the analysis files to be used in the manual coupling. This process produces the following files:

- the interface file to be adapted and manually inserted into the target input deck,
- the `.mat` file corresponding to the Digimat material used in the analysis,
- a copy of relevant manufacturing data files.

In the particular case of a Nastran SOL1XX model coupled with viscoelastic Digimat materials, the user is requested to provide extra information in a pop-up window before the generation of the coupled analysis files can proceed. For frequency response analyses (SOL108 and SOL111), the user first needs to decide whether or not to account for the frequency dependence of the material properties and to choose the Nastran version which is targeted (see Figure 7-1). If **No frequency dependence** is selected, the analysis will be run with constant (i.e. frequency-independent) properties corresponding to particular "reference" frequencies which can be defined independently for each viscoelastic Digimat material. If **Frequency dependence** is selected, the way the frequency dependent properties are handled depends on the Nastran version which is targeted (see section Frequency Dependent Properties).

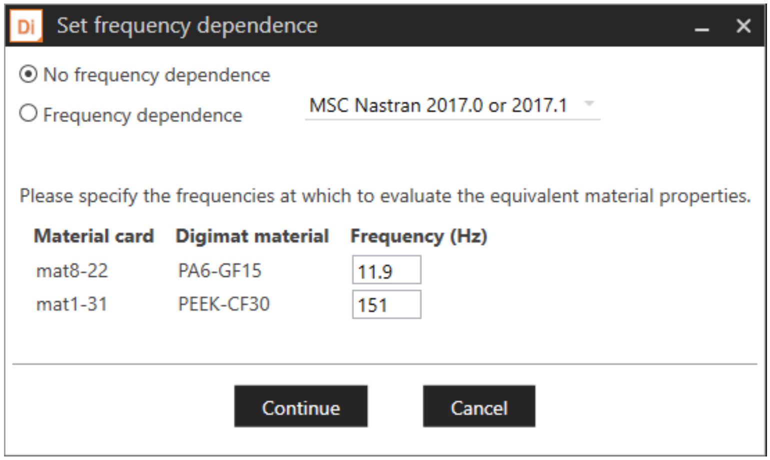

Figure 7-1 Refer to the image above: Additional input information needed in the case of a Nastran SOL108/111 analysis with viscoelastic Digimat materials when “No frequency dependence” is selected.

- For SOL108 analyses, when targeting Nastran 2017.0 or 2017.1, a separate input deck is written by Digimat-RP for each frequency at which the response needs to be computed. For example, let us consider a Nastran SOL108 model named `model.bdf` which is meant to compute the frequency response at 3 frequencies: 4 Hz, 15 Hz and 107 Hz. If it is loaded in Digimat-RP and coupled with one viscoelastic Digimat material and if the user requests for the frequency dependence of the material properties to be taken into account, Digimat-RP will produce a total of three input decks, respectively named`model_DigimatCoupled_1_of_3.bdf`,`model_DigimatCoupled_2_of_3.bdf` and `model_DigimatCoupled_3_of_3.bdf`. Each of these will correspond to a frequency response analysis at a single frequency respectively equal to 107, 15 and 4 Hz.
- When targeting Nastran 2018.0 or a newer version, which all offer specific features to handle frequency-dependent properties, a single input deck requiring a single run is created.

In the particular case of a SOL111 analysis (modal frequency response analysis), the user still needs to define “reference” frequencies even when it is decided to account for the frequency dependence of the properties (see Figure 7-2). However, these frequencies are only used to evaluate the equivalent material properties considered when extracting the eigenmodes which will be used to decompose the solution.

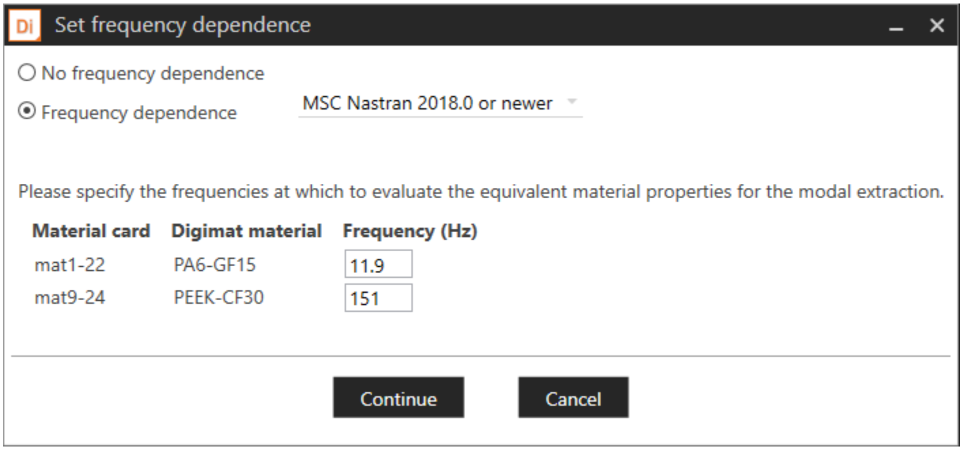

Figure 7-2 Refer to the image above: Additional input information needed in the case of a Nastran SOL111 analysis with viscoelastic Digimat materials when “Frequency dependence” is selected.

For all other linear solution sequences and regardless of the Nastran version, it is not possible to account for the frequency dependence of the properties of the viscoelastic Digimat materials. The option **Frequency dependence** is therefore grayed out and the user is requested to define the “reference” frequencies at which to evaluate these properties in the analysis (see Figure 7-3).

The generation of the material files may take some time when using Hybrid procedure due to the generation of hybrid parameters. This generation can be skipped if the parameters have already been generated before in Digimat-RP, or if they are already present in the Digimat material file. It will also be partially or completely skipped if the `.mat` file contains **encrypted**, **stiffness-locked** or **fully-locked** (see Hybrid Parameters Reverse Engineering (for Homogeneous FFF/FDM)) hybrid parameters. An example of output summary is shown in Figure 7-4. All these files are generated in a directory that can be customized at the beginning of the submission. By default, the output directory is

```
<Digimat-RP working directory>\<name of the structural model>
```

e.g.,

```
C:\MSC.Software\Digimat\working\ThreePointBendingBeam.
```

The **Export batch (RPF)** file allows you to generate a batch project file (`.rpf`) corresponding to the current project, which allows to re-run the coupling process in batch mode. More information can be found in section RP Coupling Batch Project (RPF) File.

The **Copy files** button allows you to copy the listed files to another location whereas the **Run locally** allows you to start the coupled analysis on the computer. A shortcut to the Settings manager is also present to allow you to modify the configuration before starting the analysis, if necessary. In the case of a SOL108 analysis with Nastran 2017.0 or 2017.1 accounting for the frequency dependence of the viscoelastic properties, two script files also get created and can be found alongside the input decks. One Windows batch file named `run_XXX_DigimatCoupled.bat` and one Linux shell script named `run_XXX_DigimatCoupled.sh`, where XXX is the name without any extension of the structural model which was loaded in Digimat-RP. These can be used to run the multiple single frequency jobs (or rerun some of them) sequentially on Windows and on Linux.

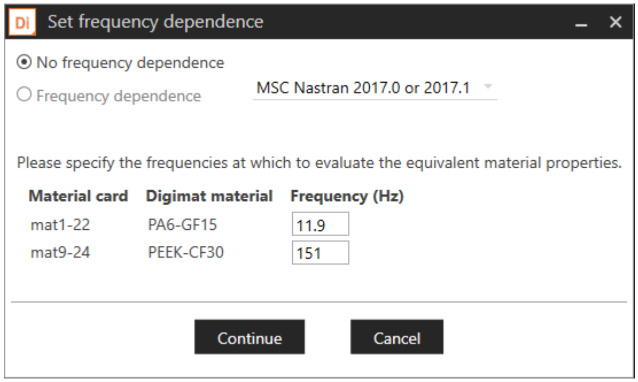

Figure 7-3 Refer to the image above: Additional input information needed in the case of one of the other Nastran SOL1XX analyses with viscoelastic Digimat materials.

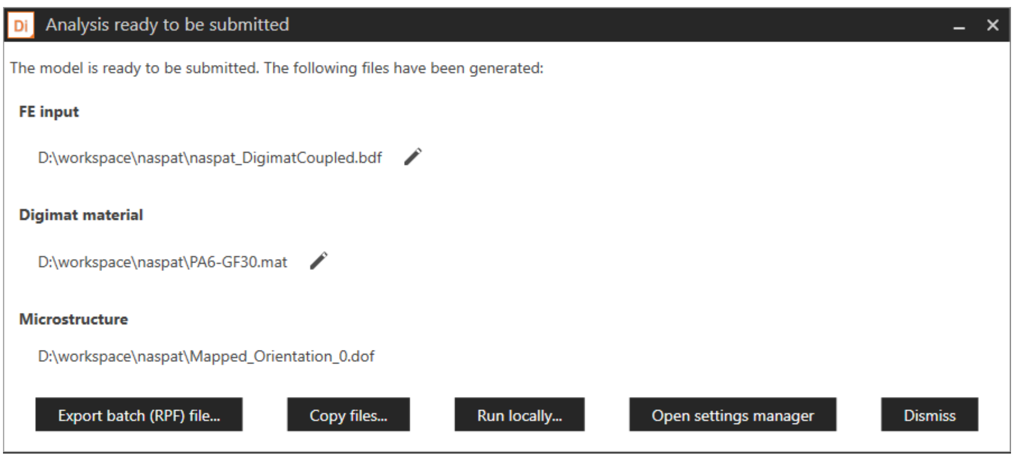

Figure 7-4 Output summary before submitting a job.

Running and Monitoring Jobs

Upon starting a local job, the window in Figure 7-5 appears. This window allows you to specify the directory where the job will run. This directory may but does not have to be the same as the directory containing the generated analysis files. The required files will be automatically copied to the specified location, it is thus not necessary to use the **Copy files** button in Figure 7-4 before starting a local job.

When a new job starts, the FE analyses manager opens up as shown in Figure 7-6. The window is divided into three panels. The upper panel contains a list of all jobs started during the current session in Digimat-RP and their status. Clicking on a job displays more information in the two other panels: the middle one contains a copy of the status file associated to the job, and the bottom one contains a copy of the console output and/or the log files produced during the run, including Digimat log files.

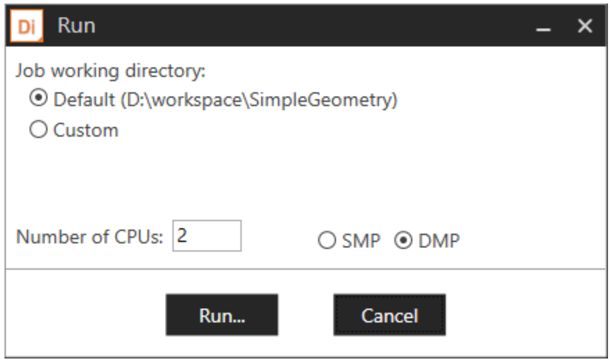

Figure 7-5 Starting a job.

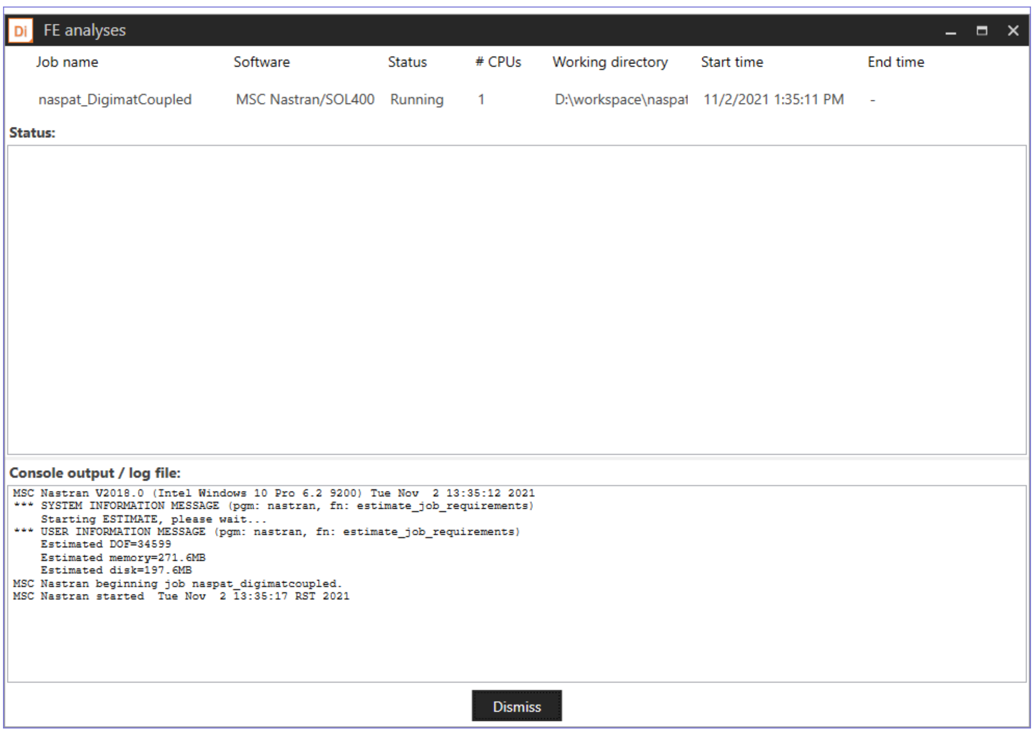

Figure 7-6 The FE analyses manager.

In the case of a SOL108 analysis with Nastran 2017.0 or 2017.1 which accounts for the frequency dependence of the viscoelastic properties, the analyses manager looks slightly different, as shown in Figure 7-7. It also includes on the left-hand side a scroll-down menu to select for which of the many single frequency jobs being run to display the console output and the log file and, on the right-hand side, a progress bar showing which of these jobs is currently running and how many have already completed and still need to run.

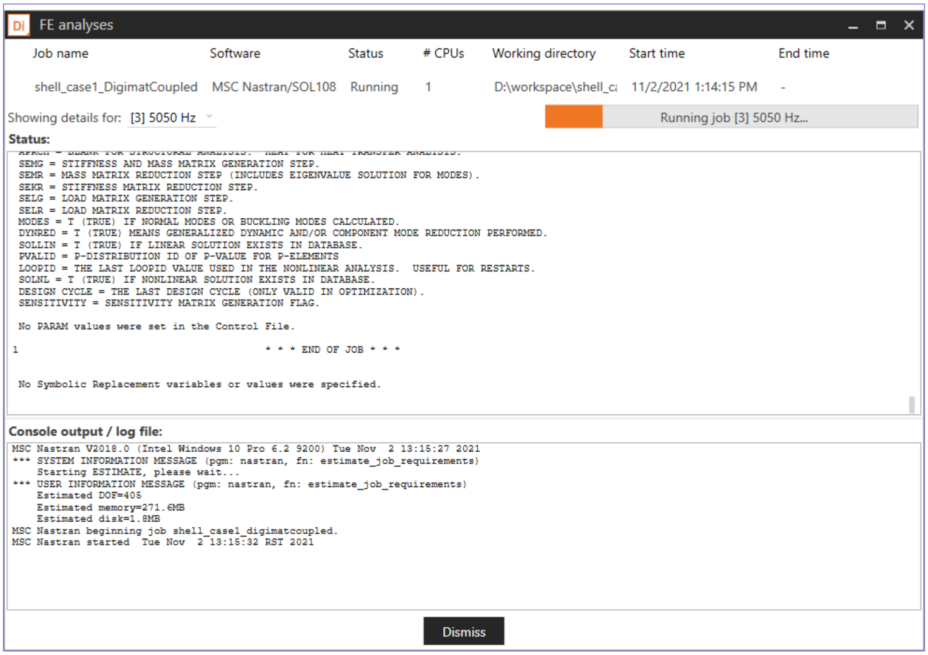

Figure 7-7 The FE analyses manager in the particular case of a SOL108 analysis targeting Nastran 2017.0 or 2017.1 which accounts for the frequency dependence of the material properties.

In the case of Run fatigue FEA from static FEA results workflow and Run static and fatigue FEA workflow, the analysis manager includes:

- on the left-hand side a scroll-down menu to select for which of the jobs being run to display the console output and the log file. There are two jobs, one is to launch the generation of the interface file, the second one is to launch CAEfatigue or nCode. In case of the second job, the output and log file will remain empty.
- on the right-hand side, a progress bar showing which of these jobs is currently running and how many have already completed and still need to run. For the second job, the completion will only be considered when CAEfatigue or nCode is closed.

After starting a job, Digimat-RP can immediately be used for another coupled analysis without interference with the running jobs. If Digimat-RP is exited, the job is no more monitored but is still running in the specified directory, and can be manually managed. A SOL108 analysis with Nastran 2017.0 or 2017.1 which accounts for the frequency dependence of the viscoelastic properties and which, hence, needs to run sequentially multiple single frequency jobs is an exception to that rule: the sequence of job executions will stop as soon as Digimat-RP is exited. To avoid that, the user is advised to copy the files locally and to launch the sequence of jobs with the help of one of the script files written by Digimat-RP as explained in section Submitting Jobs. By default, when a job (or a sequence of single frequency jobs) completes, a dialog box asking whether the results file should be opened is automatically shown. This behavior can also be modified in the Settings manager.

When right-clicking on a job in the upper panel, a context menu appears, with a list of possible actions concerning the selected job. The available actions depends on the FEA code used and the status of the job:

- Running: the job is currently in progress.
- Completed: the job has ended normally, a result file is available.
- Terminated: the job has ended due to errors, but a (possibly incomplete) result file is available.
- Failed: the job has ended due to errors, and no result file is available.

All FEA codes support the following actions:

- View results: opens the results file using the program specified in the Settings manager. Depending on the FEA code, results file may or may not be opened before the analysis finishes.
- View state variables table: displays a table containing the description of the state variables used by Digimat for the selected job (see Figure 7-8).
- Open working directory: opens the working directory of the job in Windows Explorer.
- Show command line: shows the exact command used by Digimat-RP to start the job.
- Terminate: kills the job before it completes.

Additional commands may be available, such as directly opening some files using the default text editor. Figure 7-9 and Figure 7-10 show, as an example, the available actions for an Abaqus job when it is running and at its completion.

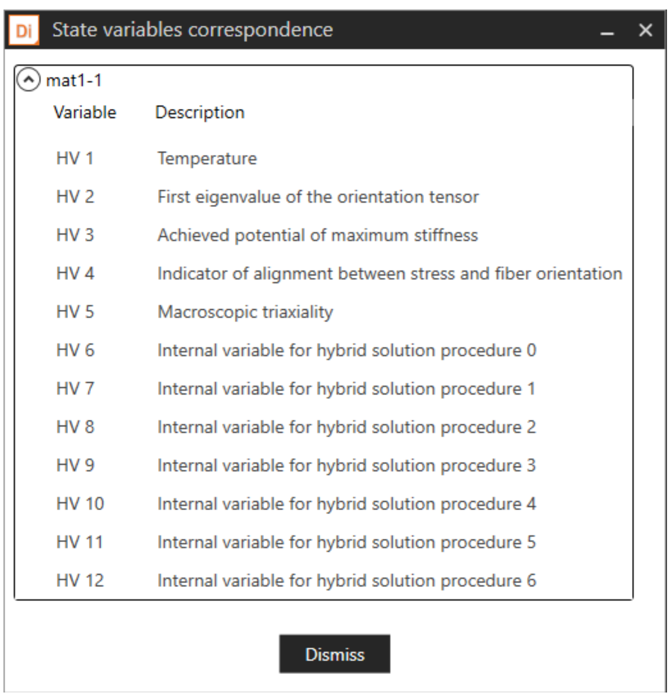

Figure 7-8 State variables used by Digimat for each Digimat material used in the selected job.

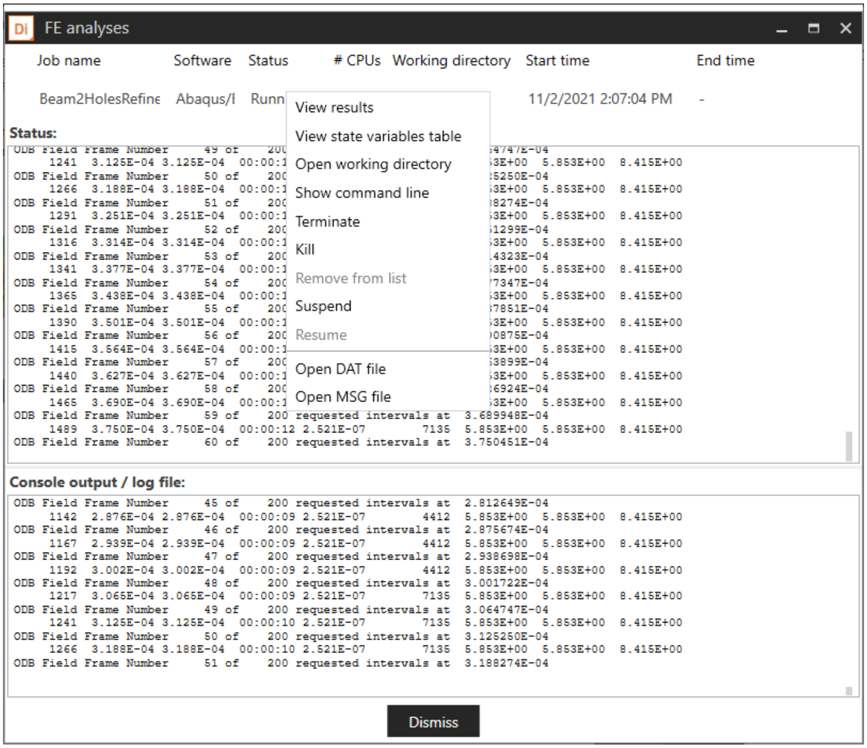

Figure 7-9 Available options when monitoring a running Abaqus job.

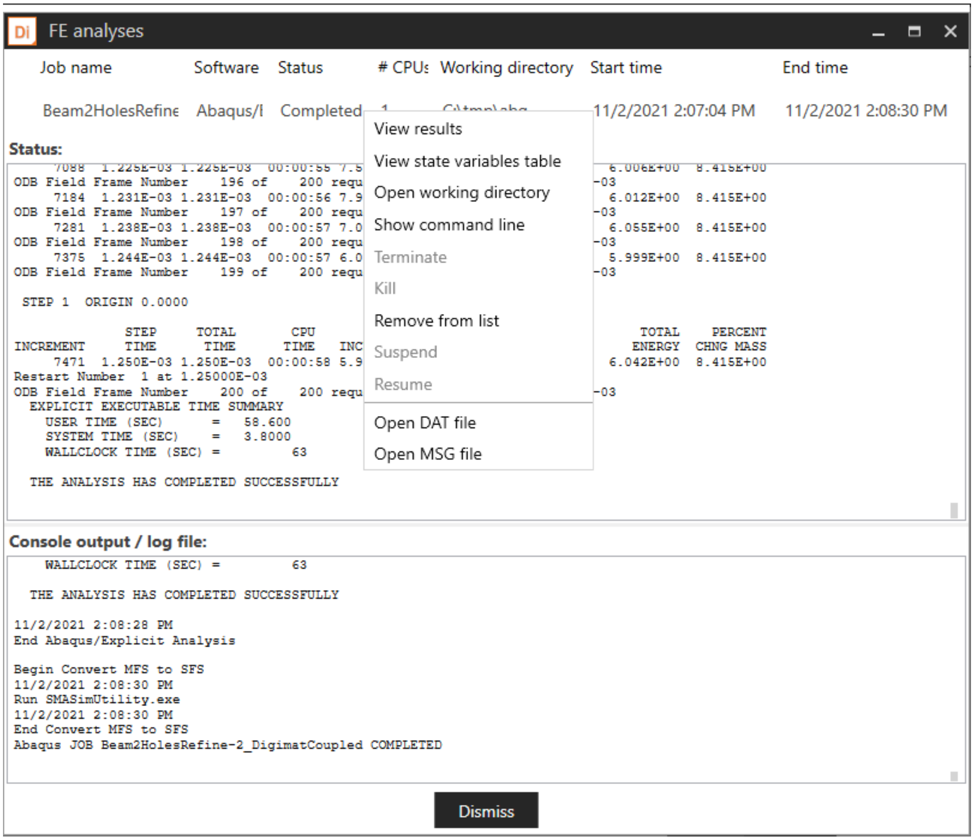

Figure 7-10 Available options at the completion of an Abaqus job.

8 Fatigue Lifetime Computation Window

- Fatigue Workflows
- Lifetime Computation Methods
- Visualization Options
- General Management

The **FATIGUE POST-PROCESSING** window (see Figure 8-1) is either the last step of the Run static and fatigue FEA or the only step of the Run fatigue FEA from static FEA results workflow when our internal Fatigue post-processing tool has been selected as Fatigue interface.

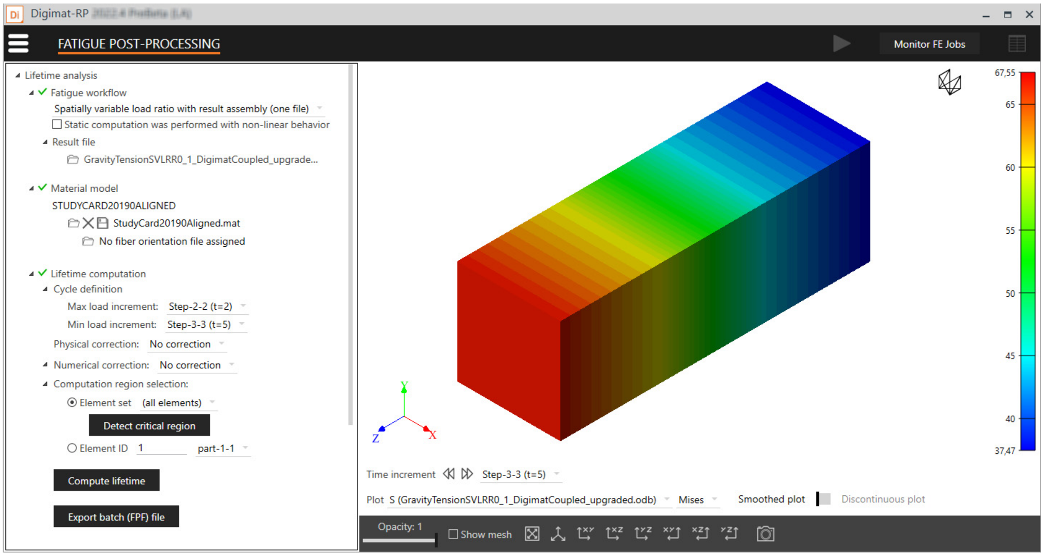

Figure 8-1 A Fatigue post-processing window at the end of the workflow.

It consists in the following areas:

- a tree view (on the left of the window) summarizing the lifetime analysis parameters. This tree view guides the user through different workflows depending on the performed FEA simulations to reach the part lifetime, e.g., minimum lifetime encountered over a part or a sub domain.
- a 3D visualization area (on the right of the window), which enables to display FEA simulation results and lifetime predictions results as field views upon the FEA model (see subsection Saving the Fatigue Post-processing Project).

The fatigue post-processing environment only supports FEA results obtained from Digimat standalone fatigue simulations at constant amplitude loading, e.g., nCode simulations are not supported yet. Supported FEA results include:

- Abaqus-Digimat results
- Ansys-Digimat results
- Marc-Digimat results
- PERMAS results

Please refer to section Fatigue Post-processing to learn more about known limitations.

Fatigue Workflows

Three different workflows are available ranging from fatigue load case screening toward design validation use.

Constant Load Ratio

The constant load ratio workflow is the most pragmatic fatigue workflow available relying on Digimat constant load ratio FEA simulations. Once the FEA result file is loaded, the user selects a material file with fatigue failure indicator. It is recommended to use the one used for FEA coupled simulation to prevent from any stiffness discrepancy. This workflow requires stress fields to recompute lifetime predictions. When fatigue failure indicator is mean stress sensitive, the load ratio can be adjusted to compute new lifetime. Within this workflow, the stresses read in FEA result file correspond to the maximum load level of the cycle, and are converted into stress amplitudes using the specified load ratio. Lifetime computation targets a specific time increment of the simulation. User can select the desired increment through a drop down list containing all identified time increments of the loaded FEA result file. The computation method options are described hereafter.

Spatially Variable Load Ratio with Result File Assembly (two files)

The second workflow consists in the assembly of two time increments to define minimum and maximum cycle timings to define a load cycle. It offers flexibility to compute lifetime in many different loading configurations (stress amplitude and load ratio) based on only monotonic loading simulations. The user is invited to load two FEA simulation result files. The two files are typically one results file going up to the maximum load level and one results file going up/down to minimum load level. Time increments of the loaded result files can be selected to define minimum and maximum load levels of a cycle. Based on those time increments, the stress amplitude and load ratio are computed for each element, allowing afterward to compute lifetime.

Spatially Variable Load Ratio with Result File Assembly (one file)

The third workflow consists in the assembly of two time increments to define minimum and maximum cycle timings to define a load cycle. It offers flexibility to compute lifetime in many different loading configurations (stress amplitude and load ratio) based on only monotonic loading simulations. The user is invited to load one FEA simulation result file. The file either contain a single load to the maximum load level for positive load ratio or a full cycle simulation passing by both the maximum load level and the minimum load level. Time increments of the loaded result files can be selected to define minimum and maximum load levels of a cycle. Based on those time increments, the stress amplitude and load ratio are computed for each element, allowing afterward to compute lifetime.

Spatially Variable Load Ratio with Full Cycle Simulation

The fourth workflow to compute lifetime relies on the simulation of a full cycle. It is considered as the most relevant since it accounts for geometrical non linearity while it tackles only one loading configuration (stress amplitude and load ratio). The maximum and minimum stresses are extracted from the loaded simulation over a given time interval. With the post-processing feature, it is possible to compute lifetime based on the resulting stress amplitude and load ratio using the different methods listed hereafter.

Lifetime Computation Methods

Based on stress fields, stress amplitude and load ratio are computed allowing to compute lifetime of each element. The lowest lifetime in the most critical area determines the part lifetime as the weakest link. When performing FEA simulations, modeling assumptions and numerical approximations such as mesh discretization lead to high local stress values. Those local extrema drive part lifetime computation as they are the hot spots.

Even though those weak spots are well captured, they appear numerically as over predicting stress and therefore under predicting part lifetime. To contain the FEA simulation lifetime under prediction, Digimat-RP provides a set of computation methods:

- Numerical correction:
  - the direct lifetime computation method (no correction)
  - the lifetime averaging method
  - the stress averaging method
  - highly stressed volume
- Physical correction:
  - the plasticity correction method

Element sets present in the results file are loaded to Digimat-RP post-processing environment, so that user can compute lifetime on a fraction of the model to save CPU time.

Element sets can also be automatically created through the **Detect critical region** option. This option allows you to define the results file from which you want to extract a new element set. You can also define the time step at which the extraction will be done. Elements satisfying some given condition will be considered as "critical elements" and added to the new element set. Two criteria are proposed to define the critical elements:

- based on Von Mises stress
- based on Number of cycles (only present if the results file contain the Number of cycles as state variable)

The exact threshold can be customized. It is also possible to only consider elements in a specified element set, instead of considering the whole model. The newly created element set can then be selected as computational region.

When using encrypted material card, the lifetime computation method section is hidden; the lifetime computation method and associated parameter(s) have carefully been identified by material supplier and do not need to be modified.

Numerical Correction

When running FEA computation, it is well known that the results will be dependent of the mesh used. This is even more true for results obtained from local field as a failure indicator or a fatigue lifetime. For a same fatigue failure criteria, a different result can be obtained at a same location in function of the mesh size at that location.

The dependency to the mesh size is the most visible for location close to a curvature. The lower the radius, the higher the mesh sensitivity. The dependency of the results to the mesh can be the main reason for discrepancy between experimental and numerical results when the critical fatigue location is close to a significantly small radius, specially when the slope of the S-N curve is flat. In that case, a small discrepancy on the stress predicted numerically can induced a discrepancy of several decades in term of lifetime.

The simplest way to observe such behavior is to perform a comparison between experimental and numerical results on notched coupons. Differences observed on notched coupons are the extreme cases. You may not observe such discrepancy on a real applications since:

- you may use a mesh which is coarse to speed up your computation;
- you may have your critical fatigue area far from a significantly small radius.

Nevertheless, Digimat offers a numerical correction to overcome the influence of the local stress evolving with the mesh size.

No Correction

This lifetime computation method without correction returns the lifetime as explained in section Fatigue Workflows. Typically, the same results can be obtained within the FEA simulation framework when using a Digimat material with pseudo-grain High Cyclic Fatigue, as detailed in subsection Aspects Common to Several Interfaces. In such case, the added value of Digimat-RP fatigue post-processing feature is to benefit from the different workflows, for example to cover different load ratios.

Lifetime Averaging

The lifetime averaging method is the simplest way to balance local extrema by applying a low-pass filter. It consists in computing lifetime of all elements according to a given loading (stress amplitude and load ratio) and to "logarithmic" average the lifetimes over a given spatial domain. The averaging spatial domain is a sphere defined by a radius. The averaging computation is element volume fraction based.

Stress Averaging

The stress averaging method is balancing local extrema at a lower level compared to the lifetime averaging method. It consists in computing stresses for given loading (stress amplitude and load ratio) and to average them over a given spatial domain with low-pass filter, prior to the lifetime computation. The averaging spatial domain is a sphere defined by a radius. The averaging computation is element volume fraction based.

Highly Stressed Volume

The highly stress volume method has the goal to compute a correction on the stress tensor based on the stress gradient. In order to prevent weakness observed with other method, this method detect automatically the volume of interest on which further computation need to be done. Such volume is called "highly stressed volume". This volume is build by comparing a given quantities between an element already inside the volume and a neighbor. If the ratio between the quantities in both elements respect the constraints on the threshold, the neighbor elements is added to the volume.

The threshold is thus one input for this method (see Eq. (8-1)). Two other inputs are numerical parameters a_1 and a_2, which link the evolution of the highly stressed volume with the evolution of the correction factor. A last numerical parameters a_3 is another threshold below which the correction factor cannot go.

$$\sigma_{eff} = \frac{\sigma_{max}}{\max(a_1 / f(HighlyStressedVolume) + a_2, a_3)} \tag{8-1}$$

This a_3 parameters can have 1 as maximum value. In that case, the correction factor can only be bigger than 1. This prevent the lifetime to decrease. We have enable the possibility to have a value lower than 1 leading to a decrease of the lifetime due to the practical constraints of the calibration of the fatigue failure criteria. When calibrating a Fatigue failure criteria, the calibration will be done on a dumbbell having a non infinite radius. On top of that, when the localization factor is used in the calibration, this factor is obtained for a given mesh size. This means that the fatigue failure criteria is by default valid for a given mesh size for a given radius. The numerical correction enables to take that into account. It is expected to see a correction factor below 1 for scenario with radius larger than the one of the dumbbell use to calibrate or for similar radius but coarser mesh huge. The a_3 parameter will be identified once all other parameters have been calibrated. It is the correction factor needed to obtain accurate results on a scenario with coarse mesh on a nearly infinite radius.

How to Use the Numerical Correction

The recommended numerical correction is the Highly stressed volume correction. It will provide the most accurate results for any mesh size and local radius.

For users that want to use simpler method like the lifetime averaging and stress averaging, a solution to calibrate the averaging radius is to perform a comparison between experimental and numerical results for several configurations. The averaging radius can then be calibrated on the configuration that yield the best comparison. The calibrate value is the one that enable to have a perfect match between experimental and numerical results on this best comparison. This enable to increase the probability to obtain conservative results on a wide range of results. The larger the amount of configuration to calibrate the averaging radius, the higher the probability to be conservative.

The calibration of the parameters of the Highly stressed volume methodology requires a dedicated workflow. The calibration requires to have experimental data for scenario where failure due to fatigue appears close to various radius. One proposition is to have notched specimens for various notched radius. It is expected that various mesh would be used in function of the notched radius to capture the radius as best as possible. It is also interesting to test these notched specimens for various load ratio and lifetime. The goal would be to capture various level of stress gradients and various level of plasticity. Testing various loading angle would be a bonus since it is expected that the correction is globally isotropic. A minimum configuration could be to test a single notched radii, to test it on two strongly different load ratio (-1 and 0.5 for example) and two different expected lifetime (like 1000 and 1000000). We acknowledge that we cannot know the lifetime in advance. It may require some experience to guess the right force to use for a given notched radii from the force used for the unnotched specimen.

The current proposed workflow to calibrate the three value of the Highly Stressed Volume is to calibrate first a_1 while keeping a_2=a_3=1. The results of the a_1 calibration would be to reach a constant discrepancy between all experimental data and the numerical data. The numerical results would then be too optimistic, specifically on the unnotched specimen. The second step would then be to calibrate a_2 to a value lower than 1 in order to remove the discrepancy between the experimental and numerical value on most of the curves. The slope of the S-N curve on notched and unnotched specimen would remain the same.

The last step is to calibrate a_3 to a value lower than 1. It can be calibrating in order to remove the biggest discrepancy remaining between experimental and numerical results. Calibrating this value should not affect the results for which a good correlation is already obtained. The reason why this parameter should be lower than 1 is explained below:

With our calibration workflow in Digimat-MX, what we compute is $S_{(\text{TypeS dumbbell})} = Kf_{(\text{Type S radius with given mesh size})} * S_{(\text{Infinite radius})}$. Since we use the localization factor which is obtained for a given mesh size, our Kf is bigger than 1, let say 1.X.

When we validate on an application, the $S_{(\text{application})} = Kf_{(\text{application})} * S_{(\text{Infinite radius})}$ can be lower than 1.X if the radius is larger than the Type $S_{\text{dumbbell radius}}$ or if the mesh is coarser than the one used to compute the localization factor. With this definition of Kf, i.e. always reported to $S_{(\text{Infinite radius})}$, Kf can indeed never be lower than 1.

But the Kf that we compute in RP is not reported to $S_{(\text{Infinite radius})}$ but to $S_{(\text{Type S dumbbell})}$. It is $Kf_{(\text{Effective})} = Kf_{(\text{Application})} / Kf_{(\text{Type S dumbbell})}$. This Kf effective should be bigger than 1/1.X and so it can go lower than 1.

Physical Correction

Plasticity Correction

Geometric discontinuities in engineering structures generate local stress levels that may exceed the yield stress in the notch area. On the other hand, the use of a linear material model for the FEA leads to artificially high stress levels and thus to an underestimation of the part's lifetime. The plasticity correction method allows to capture local plasticity and to convert elastic stress and strains into plastic stress and strains when plastic deformation occurs. The method is based on an equivalence between the strain energy computed in the plastic zone from elastic FEA and the one using an elastic-plastic material model. Hence, this method allows to take advantage from the low computational cost of linear FEA while taking into account the plasticity in a post-processing step. Two options are available for this correction method:

- Strain based: the given elastic solution and its elastic-plastic counterpart have proportional strains.

- Stress based: the given elastic solution and its elastic-plastic counterpart have proportional stresses.

If the plasticity correction method is chosen then the user needs to select, in addition to the elastic material card that was used for the FEA, a second material card with the plastic parameters calibrated as for a quasi-static analysis and the associated hybrid parameters. The GUI checks the compatibility of the material's properties from the two cards, i.e., the density, the Young's modulus and the Poisson's ratio should be the same in both cards, also the microstructure parameters namely the mass fraction, the volume fraction and the aspect ratio of the inclusion phase should be the same. This correction method could be used alone or combined with one of the numerical corrections described above. If a combination of the corrections is chosen, then the numerical correction applies to the plastically corrected stress.

Visualization Options

The goal of these options is to help each customer to visualize the critical area and to focus on the elements for which a Digimat material is assigned.

Filtering

The filtering option allow each customer to only visualize the elements for which a Digimat material has been assigned in the Material Model section of this tool.

Cutting planes

This cutting planes option enables you to cut the mesh by giving min and max coordinates in the three axis directions.

Highlighting

This highlighting option allow you to highlight the currently selected element set for the lifetime computation.

General Management

Saving the Fatigue Post-processing Project

The settings defined in the Fatigue post-processing window can be saved to a project file and re-loaded in a later fatigue post-processing session. Like for a regular Digimat-RP project, fatigue post-processing project can be saved in two formats:

- The Digimat complete project (`.dcp`) format, which contains a copy of the full fatigue post-processing project. Hence, the input files (FEA result files, material card, manufacturing data files...) used to create the fatigue post-processing project are not required when loading a `.dcp` file. However, the size of the `.dcp` file can quickly grow up with the size of the corresponding FEA result file.
- The Digimat lightweight project (`.dlp`) format, which is usually much smaller than the corresponding `.dcp` file. However, in order to be able to successfully load the project, the input files used to create the project must still be present at the same original location on the disk; if the input files were moved, a user interaction will be required to update these inputs location.

Moreover, opening a fatigue post-processing project from the main Digimat-RP window will automatically open the Fatigue post-processing window. This is also applies to file association in Windows: double-clicking such a `.dcp` (or `.dlp`) file will automatically start Digimat-RP then open the project in the Fatigue post-processing window.

Saving the Material Card

After having identified the appropriate lifetime computation method and associated parameter(s), the user can save the material card so that it now contains the post-processing parameters. This allows to use the same set of parameters for next post-processing projects.

Saving the Lifetime Field Results

After having performed a successful lifetime computation, the user can save the lifetime fields values (e.g. element values) for further manual post-processing, by clicking on the "Export lifetime field results" then selecting an output folder and filename. This field will be formatted as a Digimat-XML file by default (although a batch mode option exists to export to Moldflow3D format).

This file will contain, for each element being assigned a Digimat material, the corrected lifetime value. If the value is not computed or not available for a given element, a NaN (not-a-number) value is written.

Remark: If the model contains several Digimat material assignments, Digimat-RP will export one file per material assignement, using the requested filename with a suffix to identify each assignement.

3D Visualization Settings

The 3D visualization area allows to visualize the FEA model and to plot field results on the model. It features the following items:

- A visualization area where FEA results can be displayed (typically as contour plot) upon the model, and which also features an axis system triad and a results scale. The scale range can be modified by clicking on the scale. This scale is logarithmic when plotting lifetime as "number of cycles", and is linear for any other field.
- A drop down list (as well as **backward** and **forward** buttons) to select which simulation time increment is used for plotting.

- A drop down box to select which quantity field to plot. Available quantity fields are the FEA simulation output variables (loaded with the model) as well as the results of the fatigue post-processing simulation (e.g. "number of cycles" and "processed stress"), once completed.
- Options related to field interpolation for the contour plot. Fields that are not constant per-element can be plotted as average over integration points, minimum or maximum integration points; lifetime (number of cycles) is by default plotted as minimum integration point. Moreover, most fields can be displayed as a smoothed contour plot, or as discontinuous values per element.
- Graphical items to change the model opacity, show or hide the mesh (element edges), and apply predefined views.

9 Command Line/Batch Mode

- General Remarks
- Digimat Coupling
- Fatigue Lifetime Computation

Digimat-RP currently offers two workflows that can be done in command line (batch) mode: the Digimat coupling process (main workflow) and the fatigue lifetime computation. When running Digimat-RP in batch mode, the GUI is not shown and the whole workflow is done following the parameters defined in a batch project file without user interaction. This feature is well-suited for automation, e.g., design optimization, sensibility analysis or other automated comparisons. The batch mode is using the same executable as the interactive mode, and is thus subjected to the same software requirements as the interactive mode.

General Remarks

Batch project files such as RPF and FPF files are ASCII files containing several sections using the following format:

```
###########################################
SECTION NAME
parameter 1 name = parameter 1 value
parameter 2 name = parameter 2 value
...
```

The order of the parameters inside a given section is not important but a given parameter can only be defined once per section.

Some sections are named, i.e., they have a parameter name so that they can be referred to in other sections. Section names should be unique.

Parameter values corresponding to paths to input files (e.g., the FEA input/result files or the Digimat material files). These paths can be given as absolute or relative paths. If relative paths are used, the paths are assumed to be relative to the directory containing the batch file.

All lines starting with # are comments.

Digimat Coupling

RP Coupling Batch Project (RPF) File

The main Digimat coupling workflow can be performed in batch mode (without a GUI) using a simple ASCII file describing all inputs and parameters to use. This input file has a default extension of `.rpf` and can be exported from the GUI using the **Export batch (RPF) file** button (see Figure 9-1). Note that this workflow optionally includes the local submission of the coupled analysis. When exporting the `.rpf` file, a directory next to it may be created (e.g., directory `batch_files` when using `batch.rpf`). This directory contains all data files created and used during the workflow. For example, if orientation tensors were mapped from a manufacturing mesh to the structural mesh, a file containing the mapped orientation tensors will be created. Hence, it will be possible to directly use this file instead of restarting the mapping operation when performing the same workflow in batch mode.

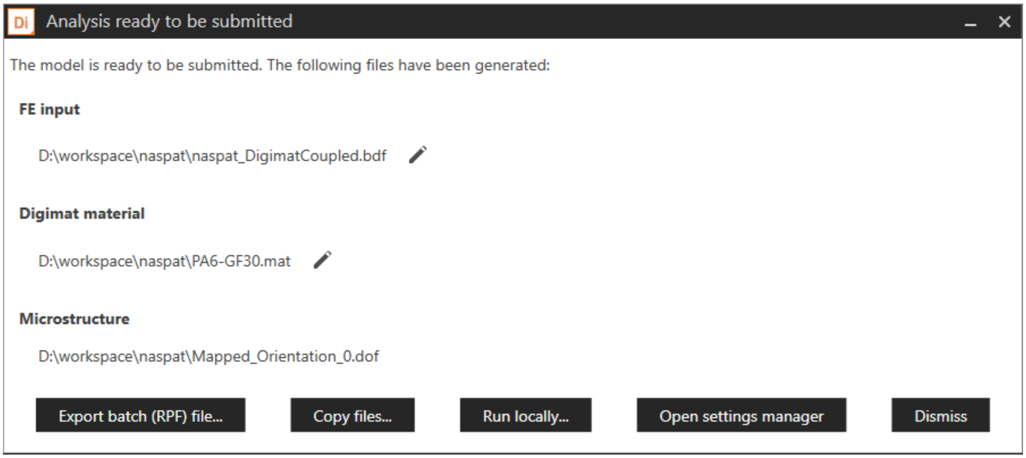

Figure 9-1 Output summary before submitting a job.

Digimat Coupling Batch Run

A batch run of the coupling process in Digimat-RP is started from the command line using

```
DIGIMAT_INSTALL_DIR\DigimatRP\exec\DigimatRP.bat batch=on
rpf="C:\Path\To\file.rpf"
log="C:\Path\To\file.log"
```

- The parameter `batch=on` (or `b=on` as short alias) is required to enable the batch mode.
- The parameter `rpf=...` is required and corresponds to the RPF file to use for the batch run. If a relative path is given, it is assumed to be relative to the current directory.
- The parameter `log=...` (or `l=...` as short alias) is optional and corresponds to the name of the log file where all messages will be written. If a relative path is given, it is assumed to be relative to the current directory. It is highly recommended to define a log file.

Starting from a Digimat Project or from a Structural Model

The batch mode can be performed starting from a Digimat project file (`*.dcp or *.dlp`). In this scenario, the batch file defines a PROJECT section with a parameter named `project_file` pointing to the location of the project file.

When starting from a structural model, the batch file defines a STRUCTURAL MODEL section containing data related to the model.

These two scenarios are mutually exclusive since only one structural model can be used at any time.

Selecting Components

In the Digimat-RP GUI, components are selected by double-clicking on the region in the 3D view, or by selecting them from the list of all components found in the model. In batch mode, the component is referred by its name, as displayed in the list of components in the GUI. The name format depends on the FEA code and is mainly based on the name, type or ID of the native material associated to the component. The name is defined in the `.rpf` file using the parameter name in the COMPONENT section. It is possible to work with several components by defining several COMPONENT sections. Note that if the provided name is not found in the list of components, Digimat-RP will automatically attempt to find an available component whose name contains the provided name. This behavior can be used to quickly define components using shortcut names.

For each component, the Digimat material parameters are defined using a named MATERIAL section, linked through the `digimat_material_parameters` parameter. The same method is used for the manufacturing process parameters, the solution settings parameters,... It is also possible to assign the same parameters to several components by linking the same MATERIAL, ... section to these components.

When several materials have to be assigned to the same component (for example in CFF), the syntax is to use a comma-separated list of section names, e.g., `digimat_material_parameters = material_section1, material_section2` and similarly for the solution settings.

Switching Data Sources

When exporting a batch file after using advanced workflows, both the original inputs and the processed data are exported and can be used in the batch run. More precisely:

- When using reverse engineering to define the Digimat material, the batch file will set the Digimat material to be loaded from a `.mat` file containing the results of the reverse engineering. The parameters related to the reverse engineering are also written in the `.rpf` file. To perform the full reverse engineering workflow starting from the experimental data, change the value of the parameter `digimat_material_source` from file to `reverse_engineering`.
- When using fields mapping, the batch file will set the manufacturing data to be loaded from files containing the results of the mapping (if this is allowed). The parameters related to the manufacturing mesh and data are also exported. To perform the full mapping workflow, change the value of the parameter `manufacturing_data_source` from `mapped_files` to `simulation_results`.
- If an automatic superposition has been done for the fields mapping, the resulting transformation will be written in the `.rpf` file. To recompute an automatic superposition, change the value of the parameter `mapping_superposition` from file to auto.
- When using injection simulation, the batch file will set the manufacturing data to be loaded from files containing the results of the injection simulation mapped on the structural model (if this is allowed). The parameters related to the injection simulation are also written. To perform the full injection simulation workflow, change the value of the parameter `manufacturing_data_source` from `mapped_files` to `injection_estimation`.

- When using injection simulation workflow, it is possible to reuse the results of the simulation (fiber orientation, weld lines) and only redo the data mapping towards the structural model. In this case, change the value of the parameter `estimation_run_simulation` from yes to no.
- When using Hybrid solution, it is possible to force the recomputation of hybrid parameters instead of using the already available parameters. In this case, change the value of the parameter `recompute_- hybrid_parameters` from no to yes.
- After creating the coupled analysis files, the job is automatically submitted by default. To only generate the coupled files, change the value of the parameter `run_fea_job` from yes to no.

Additional Remarks

- The parameter `project_directory` specifies the location where Digimat-RP will write the coupled analysis files. In a standard usage, this directory should not contain a `DIGIMAT_Settings.ini` file. Such file will be automatically deleted to avoid conflicts. In order to keep and use such file, change the value of the parameter `delete_local_settings` from yes to no.
- The parameter `fea_job_directory` specifies the location used to run the coupled analysis.
- When using injection simulation, the parameter `estimation_working_directory` specifies the location used to run the simulation.
- If the FEA analysis is to be automatically started by Digimat-RP, the required paths related to the FEA software should be properly defined in the `.rpf` file. The parameter `wait_for_fea_job_completion` determines whether Digimat-RP should automatically exit immediately after submitting the job, or only after the job has finished running.

Fatigue Lifetime Computation

Fatigue Post-processing Batch Project (FPF) File

The fatigue lifetime computation workflow can be performed in batch mode (without a GUI) using a simple ASCII file describing all inputs and parameters to use. This input file has a default extension of `.fpf` and can be exported from the GUI using the **Export batch (FPF) file** button (see Figure 9-2). This allows running larger campaigns containing several lifetime computations without user interaction.

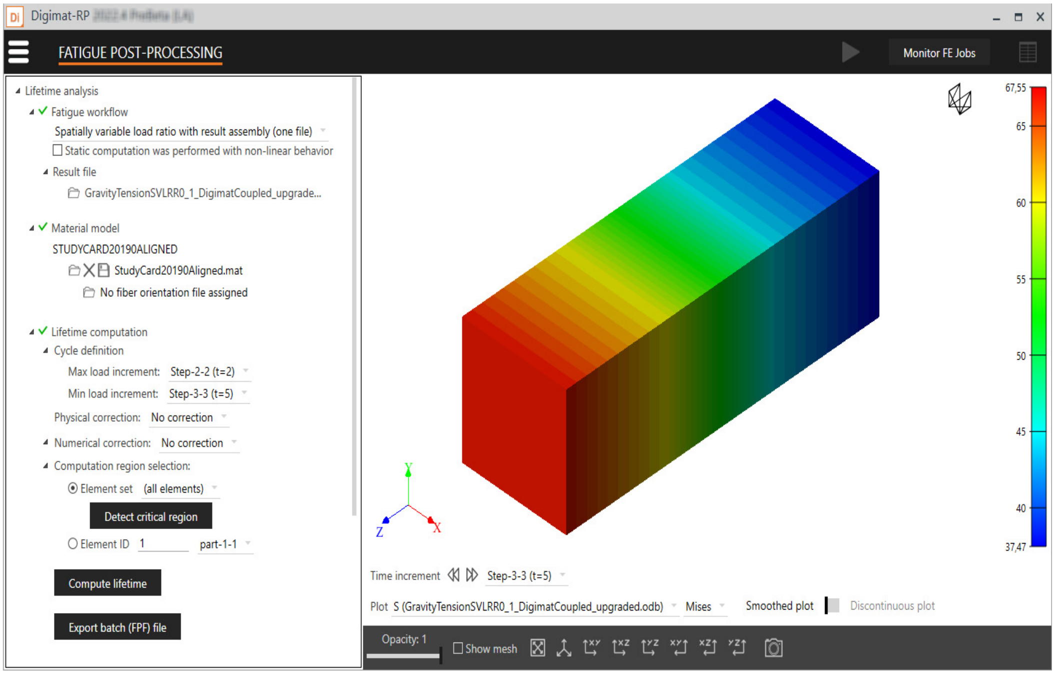

Figure 9-2 The Fatigue post-processing window.

Fatigue Post-processing Batch Run

A batch run of lifetime computation in Digimat-RP is started from the command line using

```
DIGIMAT_INSTALL_DIR\DigimatRP\exec\DigimatRP.bat batch=on
fpf="C:\Path\To\file.fpf"
log="C:\Path\To\file.log"
```

- The parameter batch=on (or b=on as short alias) is required to enable the batch mode.
- The parameter fpf=... is required and corresponds to the FPF file to use for the batch run. If a relative path is given, it is assumed to be relative to the current directory.
- The parameter log=... (or l=... as short alias) is optional and corresponds to the name of the log file where all messages will be written. If a relative path is given, it is assumed to be relative to the current directory. It is highly recommended to define a log file.

Fatigue Post-processing Batch Run Results

The location of the results file is determined by the parameter `lifetime_results_file` in the LIFETIME RESULTS section. This results file will contain the corrected lifetime (number of cycles) corresponding to the batch run.

If required, the location of the lifetime fields result file is determined by the parameter `lifetime_fields_file` in the `LIFETIME RESULTS` section; if the keyword is not specified, no such file will be generated. This field will contains the field values the corrected lifetime (number of cycles) corresponding to the batch run. It is formatted as a Digimat-XML field by default; however this format can be changed by using the keyword `lifetime_fields_format` (DigimatXML or Moldlflow3D) in the `LIFETIME RESULTS` section.

Remark: Like for interactive mode, if the model contains several Digimat material assignments, Digimat-RP will export one file per material assignment, using the requested filename with a suffix to identify each assignment.

10 Specific Features

- Additional Numerical Post-processing in Fatigue Post-processing

Additional Numerical Post-processing in Fatigue Post-processing

This development aims at supporting additional numerical correction methods for fatigue post-processing:

- Strain based gradient
- Lifetime gradient
- Stress gradient

Strain Based Gradient

The strain based gradient numerical correction uses the same type of formula as the stress based gradient described in The stress averaging method is balancing local extrema at a lower level compared to the lifetime averaging method. It consists in computing stresses for given loading (stress amplitude and load ratio) and to average them over a given spatial domain with low-pass filter, prior to the lifetime computation. The averaging spatial domain is a sphere defined by a radius. The averaging computation is element volume fraction based. The tangent option is enforced. The only change compared to the stress based gradient is that the local maximum principal stress profile derivative is replaced by the local maximum principal strain profile derivative. The correction is then directly applied to the stress tensor.

$$\sigma_{eff} = \frac{\sigma_{max}}{1 + \sqrt{\frac{L}{\varepsilon_{max}} \frac{d\varepsilon}{dx}}} \qquad (10\text{-}1)$$

The advantage of this correction is that the strain profile is only influenced by the strain gradient and not also by the variation in the local orientation tensor, as it is the case for the stress based gradient. This correction is in limited availability because it has only validated on a single application and it has less theoretical background than its stress based gradient counterpart.

Lifetime Gradient

The lifetime gradient numerical correction uses the same type of formula as the stress based gradient described in The stress averaging method is balancing local extrema at a lower level compared to the lifetime averaging method. It consists in computing stresses for given loading (stress amplitude and load ratio) and to average them over a given spatial domain with low-pass filter, prior to the lifetime computation. The averaging spatial domain is a sphere defined by a radius. The averaging computation is element volume fraction based. The tangent option is enforced, and the gradient direction can only be "custom" (user-defined).

Another change compared to the stress based gradient is that the local maximum principal stress profile derivative is replaced by a lifetime-based profile derivative. More specifically, when computing the stress correction factor, the stress tensor σ is replaced by $\frac{1}{\log(N_c)}$ (and not N_c). The correction is then directly applied to the stress tensor.

$$\sigma_{eff} = \frac{\sigma_{max}}{1 + \sqrt{\frac{L}{\frac{1}{\log((N_c)max)}} \frac{d\frac{1}{\log((N_c)max)}}{dx}}} \tag{10-2}$$

The advantage of this correction is that the lifetime profile is directly influenced by the anisotropic behavior, instead of working with the maximum principal stress for the stress gradient based numerical correction. This correction is in limited availability because it has only validated on a single application and it has less theoretical background that its stress based gradient counterpart. A known weakness is that the lifetime gradient does not provide any correction when the initial lifetime is 1 anywhere within the profile used to evaluate the correction.

Stress Gradient Methods

The stress gradient method balances local stress extrema in the considered most critical direction depending on the gradient from the considered hot spot.

A first step consists in identifying the maximum principal stress direction and to build the stress profile with respect to a second direction. This second direction can be either:

- the intermediate principal stress direction (default),
- the minimum principal stress direction,
- or a user defined direction, which is imposed to all post-processed elements.

In a second step, this stress profile is used to compute a reduced stress state according to one of the following methods:

- The interpolation method uses the stress state at a given critical distance from the considered hot spot.
- The linear averaging method uses the averaged stress state over a given critical length from the considered hot spot.
- The tangent method accounts for the local maximum principal stress profile derivative with respect to the distance over the selected evaluation direction to compute a stress tensor scaling factor.

The three options are illustrated on the Figure 10-1.

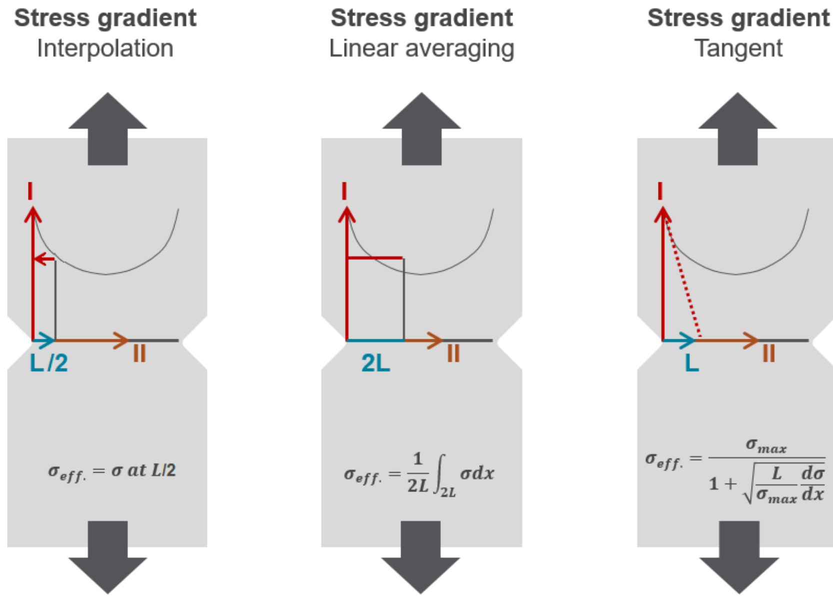

Figure 10-1   Stress gradient post-processing methods.

11 Known Limitations

- Structural Models
- Digimat Materials
- FEA jobs Submission and Monitoring
- Fatigue Post-processing
- Remarks

Structural Models

General

- Input decks containing the same material assigned to shell and solid elements are not supported for the assignation of a Digimat material. However, it is possible to assign the same Digimat material to two regions associated to different materials in the input deck.
- Entities such as nodes, elements, etc. should be explicitly defined in the input deck, i.e., with constant numerical values. For example, definitions using loop structures, mathematical operations, functions or modifications of previous definitions are not supported.
- All nodes in the input deck must have distinct ids. All elements in the input deck must have distinct ids. This must also hold across different regions/parts/instances.
- References to variable parameters such as *PARAMETER cards in Abaqus or LS-DYNA are not supported.
- Multilayer shell sections/parts must have the same material assigned to each layer to allow the assignation of a Digimat material. Materials assigned to layers of such non-homogeneous composite sections/parts cannot be assigned a Digimat material.
- Customized unit systems are not supported.
- Encrypted or zipped data in input deck are not supported.
- Abaqus: initial stresses are supported only if the input deck is flat.
- Gauss and Lobatto integration rules are available only for LS-DYNA.
- CFRP process: Only Abaqus, ANSYS and Marc are supported. In particular, LS-DYNA, MSC Nastran, PAM-CRASH and Samcef are not supported for the CFRP workflow.
- CFRP process: When using CFRP process, the regions of the structural FEA model where a unidirectional or woven-reinforced Digimat material is to be used should have an adequate coordinate system and/or orientation pre-defined in the original structural FEA model.
- CFF process: Only Abaqus, ANSYS and Marc are currently supported in Digimat-RP. Models using other formats must be coupled manually with Digimat using Digimat-MAP and Digimat-CAE.

Included Files

- MSC Nastran: only the simple include 'file.dat' command is supported. The file name/path must be explicit and be single-token (no include 'file' '.dat'). The quotes are optional only if the command is single-line. Other variants are not supported.
- OptiStruct: only the simple include 'file.fem' command is supported. The file name/path must be explicit and be single-token (no include 'file' '.fem'). The quotes are optional only if the command is single-line. Other variants are not supported.
- Ansys: only the two variants /input, <file>, <ext>, <folder path>, <n>, and /input, <file path>, <ext>, <n> are supported. The line number n is optional and indicates where Digimat-RP starts to read the file. Labels and /eof commands are not supported.

- LS-DYNA: only the cards `*INCLUDE`, `*INCLUDE_PATH`, `*INCLUDE_PATH_RELATIVE` and `*INCLUDE_TRANSFORM` are supported. For `*INCLUDE_TRANSFORM`, only node id offset, element id offset, part id offset and transformation id are supported. POS6P transformations are not supported.
- PERMAS: paths defined with the `DEFINE` command are not taken into account. All files must therefore be located in the same directory as the file they are included in or located with their absolute path. Besides, including files from within a section defining a new material is not supported.
- Samcef: includes are not supported.

Using the Macro Non-Linear Solution

For MSC Nastran SOL1XX and OptiStruct models only:

- Digimat materials will only affect the elements of the input deck to which either a PSHELL, PCOMP or PSOLID is assigned.
- Input decks with blank or default ply thickness values in the PCOMP cards are not supported.
- Input decks for frequency response analyses (SOL108 or SOL111) with multiple frequency set identification numbers (i.e. multiple FREQi cards with different identification numbers) are not supported.
- For frequency response analyses (SOL108 or SOL111), only FREQ, FREQ1 and FREQ2 cards are supported to define the range of frequencies to be analyzed. When Nastran 2018.0 or newer is specifically selected as target version, FREQ3, FREQ4 or FREQ5 cards are also allowed but only in addition to FREQ, FREQ1 and FREQ2 cards and they will be ignored by Digimat-RP.
- Input decks in which the bulk data section begins in a file which is included by another one are not supported.

For MSC Nastran SOL108/SOL111 models, when Nastran 2017.0 or 2017.1 is selected as target version:

- When requesting for the frequency dependence of the material properties to be taken into account, the `.xdb` format is the only supported output format. Digimat-RP will therefore automatically set PARAM,POST,0 in the finite element model.
- Requesting for the frequency dependence of the material properties to be taken into account when carrying out a frequency response analysis with viscoelastic Digimat materials is not allowed if the range of frequencies to be analyzed includes 0 Hz.

For MSC Nastran SOL108/SOL111 models, when Nastran 2018.0 or newer is specifically selected as target version, shell elements using viscoelastic Digimat materials will not include any shear stiffness terms.

For PERMAS models:

- Axisymmetric components (COMPONENT TYPE = AXISYM) are not supported.
- Digimat materials can only be used in replacement of homogeneous materials ($MATERIAL).
- Different components cannot reuse the same node, element, node set, element set or $GEODAT ids.

- The coordinates of the nodes of a component must be defined in the component system (which is always Cartesian).
- Components cannot be rotated and/or translated in the global reference system (i.e. $GSYS is not allowed).
- Each component can only have one structure variant ($STRUCTURE) and one system variant ($SYSTEM).
- Shell elements with a different shell thickness at each node cannot use a Digimat material.
- Shell elements QUAMS8, QUAMS9, TRIMS6, QUAM8, QUAM9, TRIM6 cannot use a Digimat material.
- If the orientation of the Digimat material is taken from the Digimat material itself (i.e. no orientation data), the corresponding orientation tensor must be diagonal.

For Abaqus models:

- Using Digimat materials with orientation file in shell sections with temperature distribution through the thickness is only supported with TEMPERATURE=1. To use temperature distribution through thickness, initial shell section must be COMPOSITE shell section with the same number of layers as in the orientation file.
- Using Digimat materials with orientation file in composite shell sections with temperature distribution through the thickness is only supported when the number of layers is the same as in the orientation file.
- For (T)EP material models, the plastic curve is only defined up to 10% of strain. This will result into a loss of accuracy beyond this limit (Abaqus will extrapolate the plastic curve as a plateau).

For LS-DYNA models:

- Using fixed orientation tensor is not supported. Only orientation files are supported.
- When using `*INCLUDE` command, only `PATH` and `PATH_RELATIVE` options are supported.
- LS-DYNA implicit is not supported.
- Gauss and Lobatto integration rules are not supported.
- `*INITIAL_STRESS` cards are not supported.
- Nodal thickness is not supported.
- Only MPP execution is supported when using Digimat-RP. SMP execution is not supported.

For ANSYS models:

- Include files are not supported in the initial input deck.
- Shells elements are not supported.
- Elements defined using the **EN** command are not taken into account (e.g. their properties are not modified).
- When thermo-mechanical Digimat materials are used, the definition of the environment temperature (e.g. the **TREF** command) is mandatory.

- The ANSYS encryption process is made by calling the ANSYS installation path specified in the `DIGIMAT_Settings.ini` file, and not the one specified in Digimat-RP. However, no differences have been observed so far between the file encrypted with the various ANSYS versions.

For all CAE codes:

- Only 2-phase material are supported.
- Microstructure is described only through orientation tensor. Variable aspect ratio, volume fraction and weldlines are not supported.
- Porosity is not supported.

Digimat Materials

General

- Only bi-phase materials are supported in injection and compression molding.
- Old-format fatigue failure indicators (prior to Digimat 2019.0) are no more supported.
- User-defined failure indicators are not supported.
- Only composite output can be selected (no advanced Digimat output management). If a material is created from a `.daf` file containing custom outputs, those custom outputs will be kept for the coupled analysis.
- When using Marc FEA solver, original Marc material cannot have cross-referenced card (like `viscelprop` or `damage` card).
- Pure thermal materials are only supported for Abaqus/Standard and Marc.
- Coupled thermomechanical materials are only supported for Abaqus and Marc.
- Materials with curing are only supported for Abaqus/Standard, ANSYS and Marc.

Using the Macro (Non-)Linear Solution

- For all FEA codes: Only bi-phase materials are supported.
- For all FEA codes except LS-DYNA: Failure criteria are not taken into account.
- For OptiStruct and PERMAS models: Only (thermo-)elastic materials are supported.
- For Nastran SOL1XX models: Only (thermo-)elastic or viscoelastic materials are supported.
- For LS-DYNA: Only elastic, elastoplastic, elasto-viscoplastic or strain-rate elastoplasticity materials are supported.

FEA jobs Submission and Monitoring

- When using a `*.cdb` model created by ANSYS Workbench, the following message may be encountered when running the job from command line or in Digimat-RP: "Shape testing revealed that X of the Y new or modified elements violate shape error limits. To review test results, please see the output file or issue the CHECK command."
  This may happen when the model contains too distorted elements. The job is still runnable in ANSYS Workbench because it disables element shape check when running a job. In order to run the job from command line or in Digimat-RP, it is possible to manually disable the check by inserting the command **SHPP,OFF** in the input deck.

Fatigue Post-processing

- Only solid elements are supported for this post-processing.
- The **constant load ratio** workflow is only supported for elastic materials.
- For the **constant load ratio** workflow, a single `.odb` allows to compute the lifetime for any load ratio only if the material contain a mean stress sensitivity.
- The **spatially variable with result file assembly** workflow is only supported for elastic material with mean stress sensitivity.
- The **spatially variable with result file assembly (two files)** workflow is not supported for Ansys result files. However the **spatially variable with result file assembly (one file)** workflow may be an alternative.
- The **spatially variable with full cycle simulation** workflow is not supported for elastic material without mean stress sensitivity.
- The **stress gradient** numerical correction is not supported for viscoelastic material.
- The **stress gradient** numerical correction is not recommended with Ansys result files; due to some specificities of the Ansys result file format, the evaluation of the stress gradient is only based on the average stress per element. The induced error is especially important with the **tangent** stress gradient method.
- The **plasticity correction** physical correction requires a second material card with the plastic parameters calibrated as for a quasi-static analysis and the associated hybrid parameters. In order to get meaningful results, the elastic and elastoplastic cards must be equivalent for all the parameters that affect the elastic behavior.

  The GUI checks the compatibility of the material's properties from the two cards, i.e., the density, the Young's modulus and the Poisson's ratio should be the same in both cards, also the microstructure parameters namely the mass fraction, the volume fraction and the aspect ratio of the inclusion phase should be the same.
- For Abaqus result files, fields with output located at both nodes and elements/integration points are not supported.
- For PERMAS results files, the input files need to be next to the results.
- PERMAS results files are only supported in `.hdf` format

- For PERMAS results files, viscoelastic material are not supported
- Result files obtained from explicit analysis (e.g. Abaqus/Explicit) are not supported.

Remarks

Some visualization instabilities may occur depending on the graphics card used. The following graphics cards have been tested successfully:

- NVIDIA Quadro K2100M
- NVIDIA Quadro K1100M
- NVIDIA Quadro K4000
- NVIDIA Quadro K4200
- NVIDIA Quadro FX1800M and FX580 and FX 3450/4000 SDI
- NVIDIA Quadro K1000M
- NVIDIA GeForce 840M
- Intel® HD Graphics 4600
- AMD Radeon HD 8790M

In case of issue, it is recommended to update the graphics card driver.

Please contact digimat.support@hexagon.com for any further information.